

LTspice IV Basic Lab Class & Getting Started Guide

Presented by:
Linear Technology FAE

Copyright © 2013 Linear Technology. All rights reserved.

The Value We Deliver

Innovative, High Performance Products

Outstanding Price/Performance Solutions

Expert Worldwide Technical Support

Best Quality and Reliability

Dependable Delivery



Dependable Delivery

- US**
- 2 locations
California
Washington
 - Specialized analog processes



2 Wafer Fabs

- Supports short and predictable lead times

Die Bank



Assembly

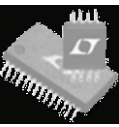
- Malaysia**
- Assembly in Malaysia
 - 90% at our facility



Test and Shipping

- Singapore**
- Completed expansion 2005

Why Use LTspice?



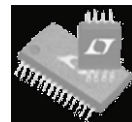
- ❖ **Stable SPICE circuit simulation with**
 - ❖ Unlimited number of nodes
 - ❖ Schematic/symbol editor
 - ❖ Waveform viewer
 - ❖ Library of passive devices
- ❖ **Fast simulation of switch mode power supplies**
 - ❖ Steady state detection
 - ❖ Turn on transient
 - ❖ Step response
 - ❖ Efficiency / power computations
- ❖ **Advanced analysis and simulation options**
 - ❖ Not covered in this lab class (sort of)
- ❖ **Outperforms or as powerful as pay-for tools**
 - ❖ In other words LTspice is free!
- ❖ **Automatically builds syntax for common tasks**

- ♦ Over 1100 macromodels of Linear Technology products
- ♦ 500+ SMPS

SPICE = Simulation
Program with Integrated
Circuit Emphasis

LTspice is also a great schematic capture / BOM tool

How Do I Get LTspice and Documentation?

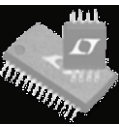


- ❖ Go to <http://www.linear.com/software>
- ❖ Left-Click on Download LTspice IV
- ❖ Follow the instructions to install

LTspice IV

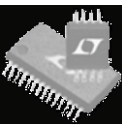
LTspice IV is a high performance Spice III simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators. Our enhancements to Spice have made simulating switching regulators extremely fast compared to normal Spice simulators, allowing the user to view waveforms for most switching regulators in just a few minutes. Included in this download are Spice, Macro Models for 80% of Linear Technology's switching regulators, over 200 op amp models, as well as resistors, transistors and MOSFET models.

- [Download LTspice IV](#) (Updated May 5, 2009)
- [LTspice Users Guide](#)
- [LTspice Getting Started Guide](#)
- [LTspice Demo Circuit Collection](#)



How Do I Get Started using LTspice?

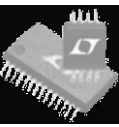
How Do I Get Started Using LTspice?



- ❖ Use one of the 100's demo circuit available on linear.com
 - ❖ Designed and Reviewed by Factory Apps Group
 - ❖ Go to <http://www.linear.com/software>
- ❖ Use a pre-drafted test fixture (JIG)
 - ❖ Provides a good starting point, but is not production-ready
 - ❖ Used to prove out part models, and are not complete designs.
 - ❖ Components are typically “ideal” components and will need to be modified based on your operating conditions
- ❖ Use the schematic editor to create your own design
 - ❖ LTspice contains models for most LTC power devices and many more
- ❖ Use simulation circuits posted on the LTspice Yahoo! User's Group.
tech.groups.yahoo.com/group/LTspice
 - ❖ Also contains many very helpful discussion threads

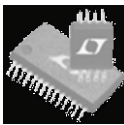
You can also check out LTspice capabilities using the education examples available on C:\Program Files\LTC\SwCADIII\examples\Educational

How Do I Get Started Using LTspice?



- ❖ Use one of the 100's demo circuit available on linear.com
 - ❖ Designed and Reviewed by Factory Apps Group
 - ❖ Go to <http://www.linear.com/software>
- ❖ Use a pre-drafted test fixture (JIG)
 - ❖ Provides a good starting point, but is not production-ready
 - ❖ Used to prove out part models, and are not complete designs.
 - ❖ Components are typically “ideal” components and will need to be modified based on your operating conditions
- ❖ Use the schematic editor to create your own design
 - ❖ LTspice contains models for most LTC power devices and many more
- ❖ Use simulation circuits posted on the LTspice Yahoo! User's Group.
tech.groups.yahoo.com/group/LTspice
 - ❖ Also contains many very helpful discussion threads

Demo Circuits on linear.com




Go to <http://www.linear.com/software>

LTspice IV

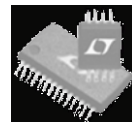
LTspice IV is a high performance Spice III simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators. Our enhancements to Spice have made simulating switching regulators extremely fast compared to normal Spice simulators, allowing the user to view waveforms for most switching regulators in just a few minutes. Included in this download are Spice, Macro Models for 80% of Linear Technology's switching regulators, over 200 op amp models, as well as resistors, transistors and MOSFET models.

- [Download LTspice IV](#) (Updated May 5, 2009)
- [LTspice Users Guide](#)
- [LTspice Getting Started Guide](#)
- [LTspice Demo Circuit Collection](#)



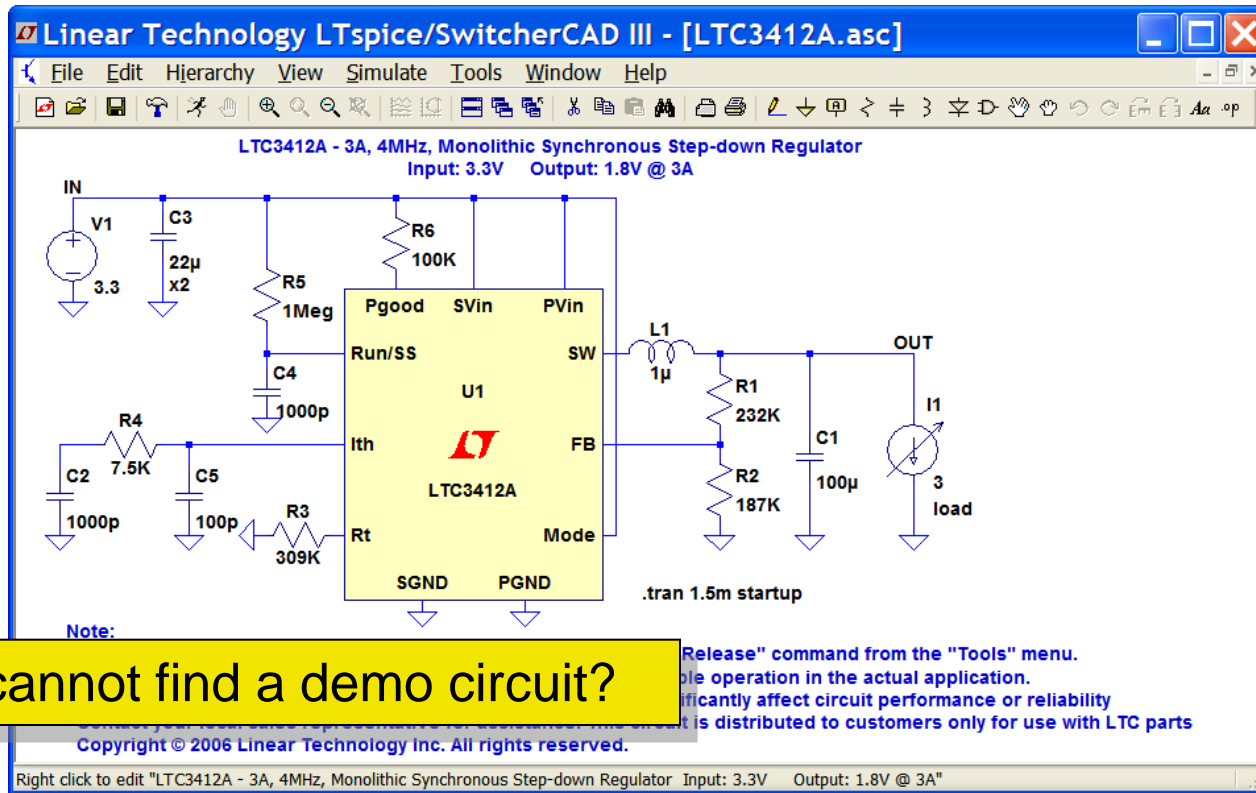
Part Number	Updated	Download
LT1071HV - 5A and 2.5A High Efficiency Switching Regulators	May 5th, 2006	LT1071HV.asc
LT1072HV - 1.25A High Efficiency Switching Regulator	May 5th, 2006	LT1072HV.asc
LT1076HV - Step-Down Switching Regulator	May 5th, 2006	LT1076HV.asc
LT1111 - Micropower DC/DC Converter Adjustable and Fixed 5V, 12V	May 26th, 2006	LT1111.asc
LT1172HV - 100kHz, 5A, 2.5A and 1.25A High Efficiency Switching Regulators	May 5th, 2006	LT1172HV.asc
LT1173 - Micropower DC/DC Converter Adjustable and Fixed 5V, 12V	Jun 12th, 2006	LT1173.asc
LT1308B - Single Cell High CurrentMicropower 600kHz Boost DC/DC Converter	May 26th, 2006	LT1308B.asc
LT1370HV - 500kHz High Efficiency 6A Switching Regulator	May 26th, 2006	LT1370HV.asc

Demo Circuits



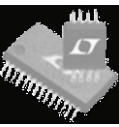
✓ Designed and reviewed by factory apps group

- ◆ It remains the customer's responsibility to verify proper and reliable operation in the actual application
- ◆ Component substitution and printed circuit board layout may significantly affect circuit performance or reliability



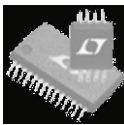
What if I cannot find a demo circuit?

How Do I Get Started Using LTspice?

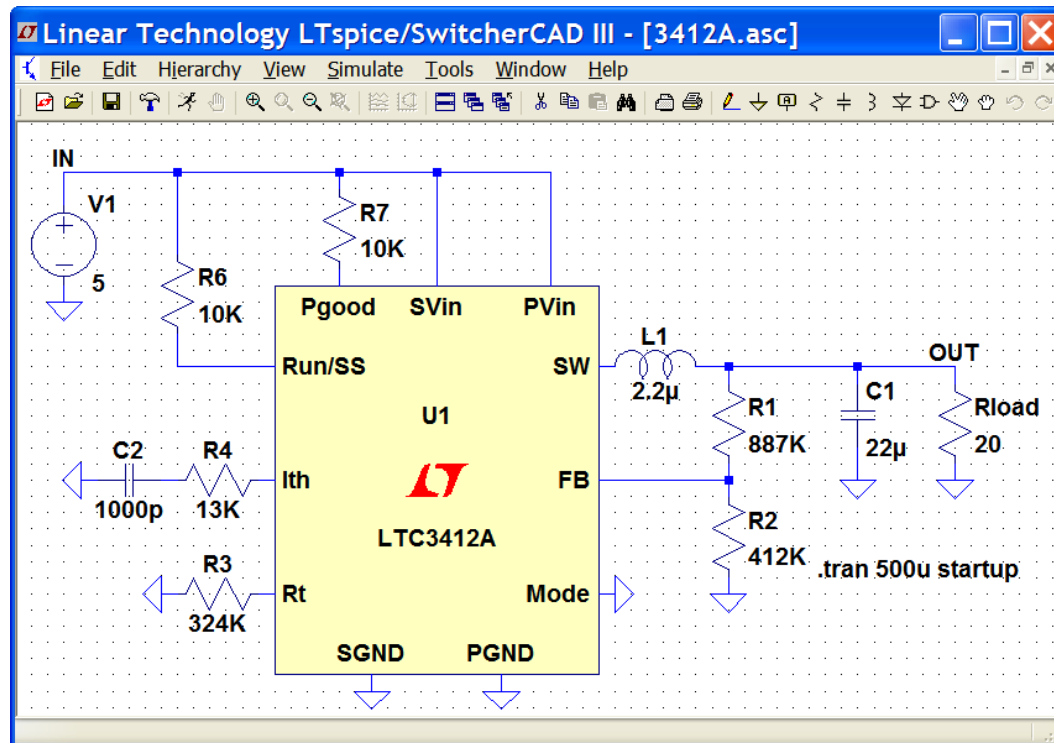


- ❖ Use one of the 100's demo circuit available on linear.com
 - ❖ Designed and Reviewed by Factory Apps Group
 - ❖ Go to <http://www.linear.com/software>
- ❖ Use a pre-drafted test fixture (JIG)
 - ❖ Provides a good starting point, but is not production-ready
 - ❖ Used to prove out part models, and are not complete designs.
 - ❖ Components are typically “ideal” components and will need to be modified based on your operating conditions
- ❖ Use the schematic editor to create your own design
 - ❖ LTspice contains models for most LTC power devices and many more
- ❖ Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
 - ❖ Also contains many very helpful discussion threads

Pre-drafted Test Fixture

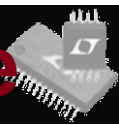


- ❖ Provides a good starting point
 - ❖ These simulations / designs are not production-ready
 - ❖ Used to prove out part models, and are not complete designs.



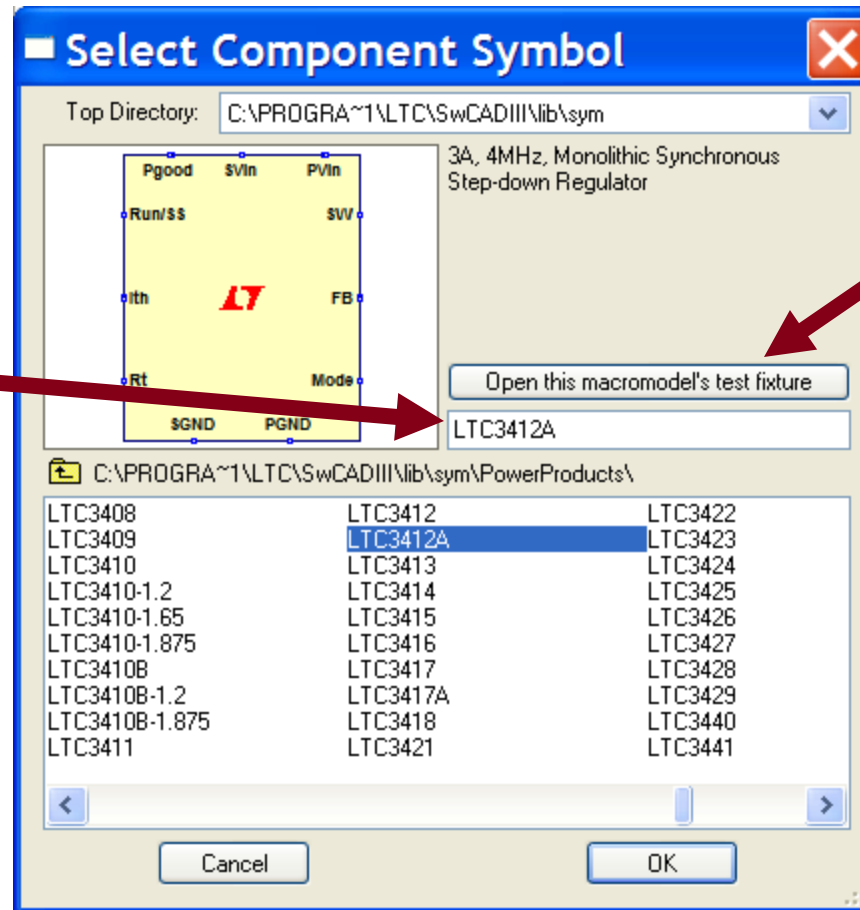
- ♦ It remains the customer's responsibility to verify proper and reliable operation in the actual application
- ♦ Printed circuit board layout may significantly affect circuit performance and reliability

Selecting a Model & Opening Test Fixture



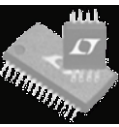
- ❖ Use the “root” part to search for the model
 - ❖ i.e. 3412A
- ❖ Select “Open this macromodel’s test fixture”

1. Enter 3412A



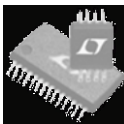
2. Select

How Do I Get Started Using LTspice?

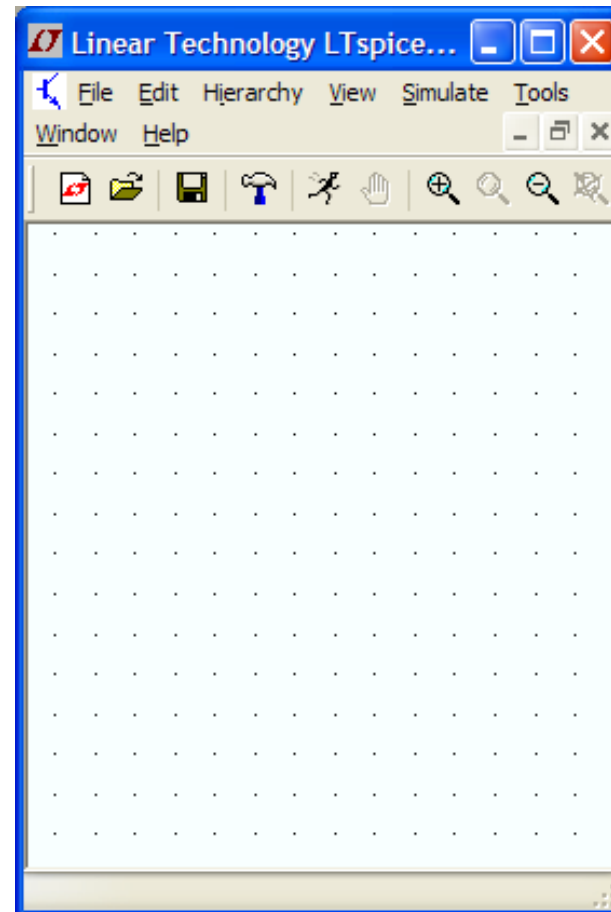
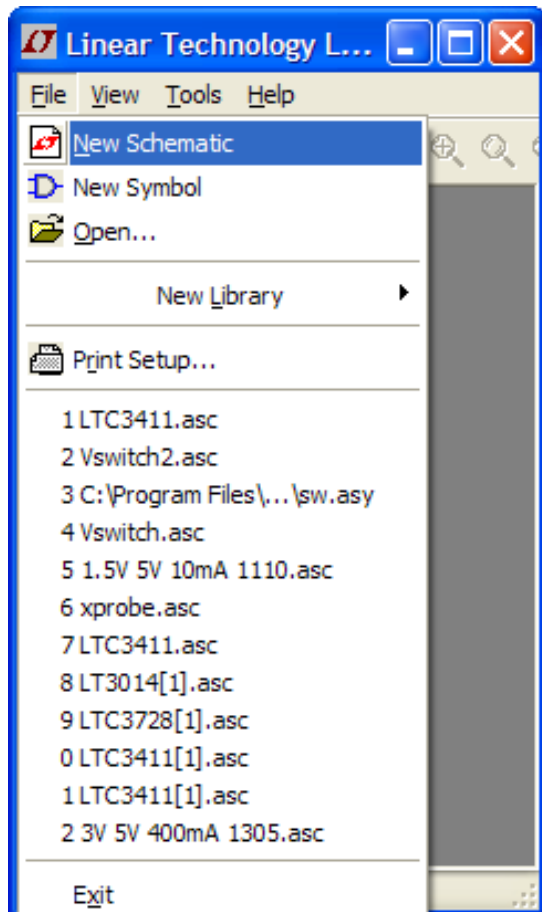


- ❖ Use one of the 100's demo circuit available on linear.com
 - ❖ Designed and Reviewed by Factory Apps Group
 - ❖ Go to <http://www.linear.com/software>
- ❖ Use a pre-drafted test fixture (JIG)
 - ❖ Provides a good starting point, but is not production-ready
 - ❖ Used to prove out part models, and are not complete designs.
 - ❖ Components are typically “ideal” components and will need to be modified based on your operating conditions
- ❖ Use the schematic editor to create your own design
 - ❖ LTspice contains models for most LTC power devices and many more
- ❖ Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
 - ❖ Also contains many very helpful discussion threads

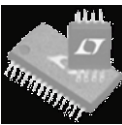
Start With a New Schematic



- ❖ Select File and New Schematic
 - ❖ Will open up a blank schematic screen



Add a Component



❖ Use Add a Component or F2

The screenshot shows the LTspice/SwitcherCAD III interface. The 'Add a Component' menu is open, showing various components like Resistor, Capacitor, Inductor, Diode, and Component (F2). The 'Component' option is highlighted. An orange arrow points to the 'Component' button on the toolbar. A red arrow points from the 'Component' menu item to the 'Select Component Symbol' dialog box. The dialog box shows a list of components organized by category, with 'FerriteBead_Z(l)' selected. A blue speech bubble at the bottom says 'Take a moment to review all of the components!'.

Linear Technology LTspice/SwitcherCAD III - [Draft2.asc]

File Edit Hierarchy View Simulate Tools Window Help

Component

Take a moment to review all of the components!

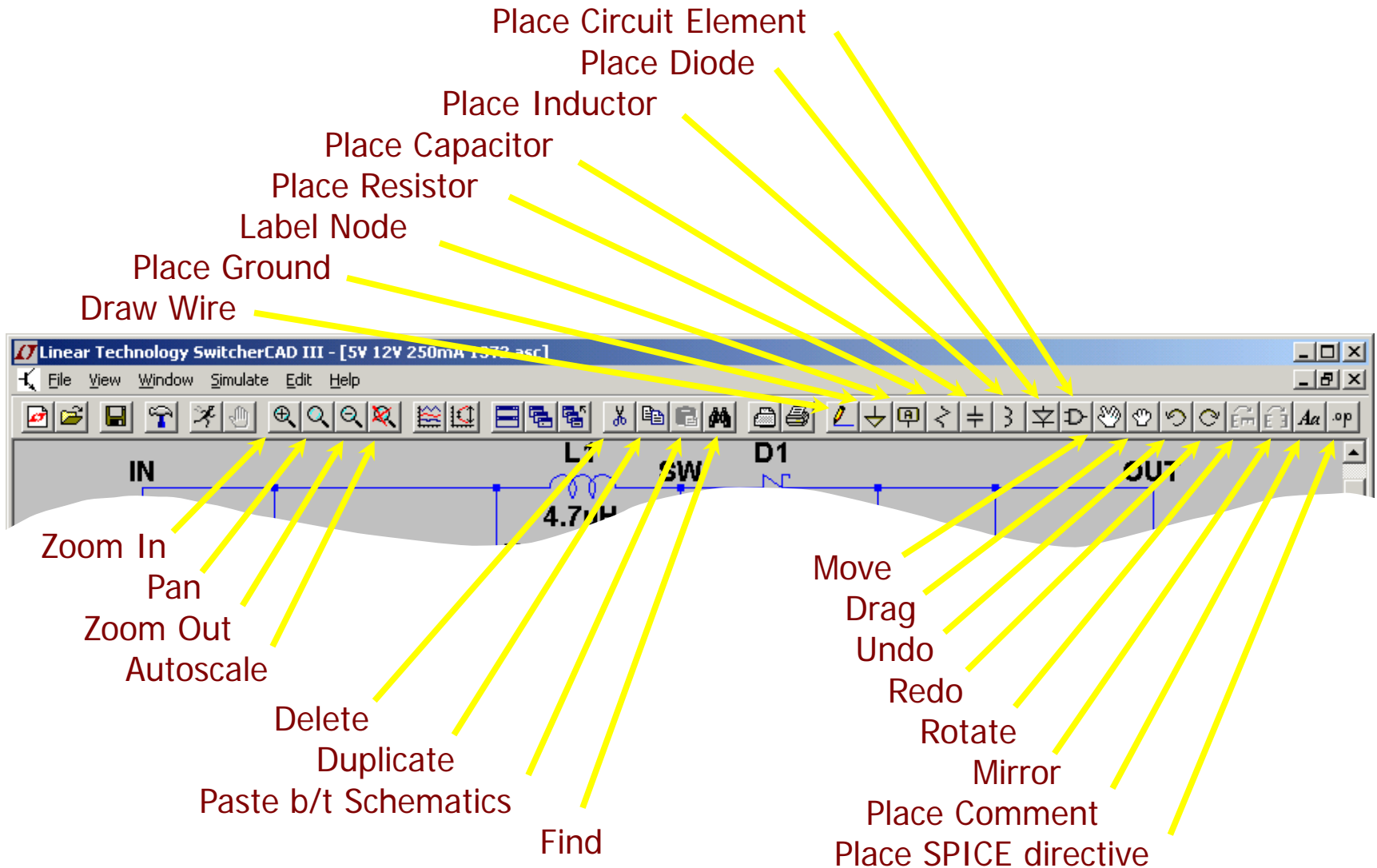
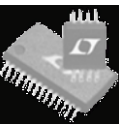
Select Component Symbol

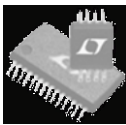
Top Directory: C:\PROGRAM~1\LTCS\SwCADIII\lib\sym

[Comparators]	bv	FerriteBead_Z(l)	mestet
[Digital]	cap	g	njt
[FilterProducts]	CNSw	g2	nmos
[Misc]	csw	h	nmos4
[Opamps]	current	ind	npn
[Optos]	diode	ind2	npn2
[PowerProducts]	e	LED	npn3
[References]	e2	load	npn4
[SpecialFunctions]	f	load2	pjt
bi	FerriteBead	lpnp	pmos
bi2	FerriteBead2	ltline	pmos4

Cancel OK

Schematic Editing





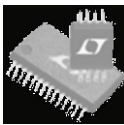
Using Labels to Specify Units for Component Attributes

- ❖ K = k = kilo = 10^3
- ❖ MEG = meg = 10^6
- ❖ G = g = giga = 10^9
- ❖ T = t = tera = 10^{12}
- ❖ M = m = milli = 10^{-3}
- ❖ U = u = micro = 10^{-6}
- ❖ N = n = nano = 10^{-9}
- ❖ P = p = pico = 10^{-12}
- ❖ F = f = femto = 10^{-15}

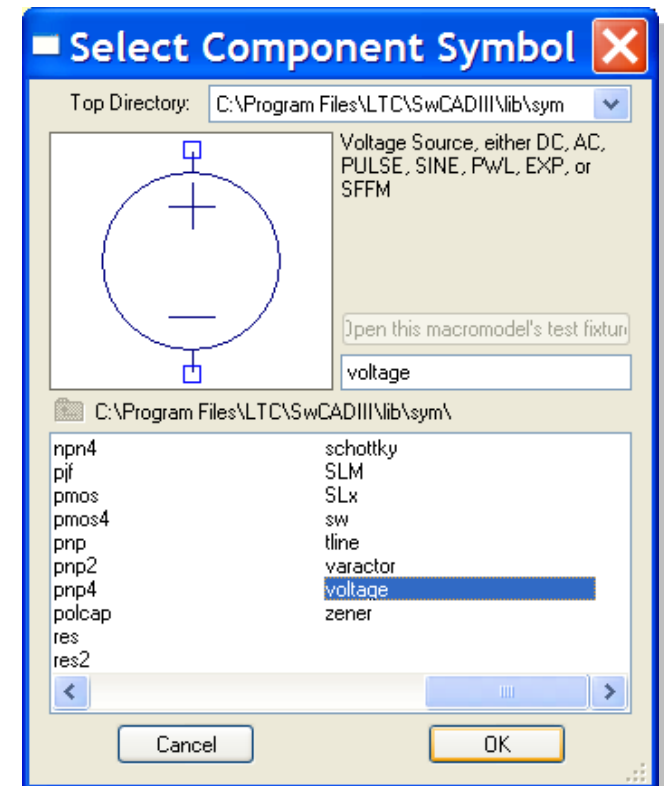
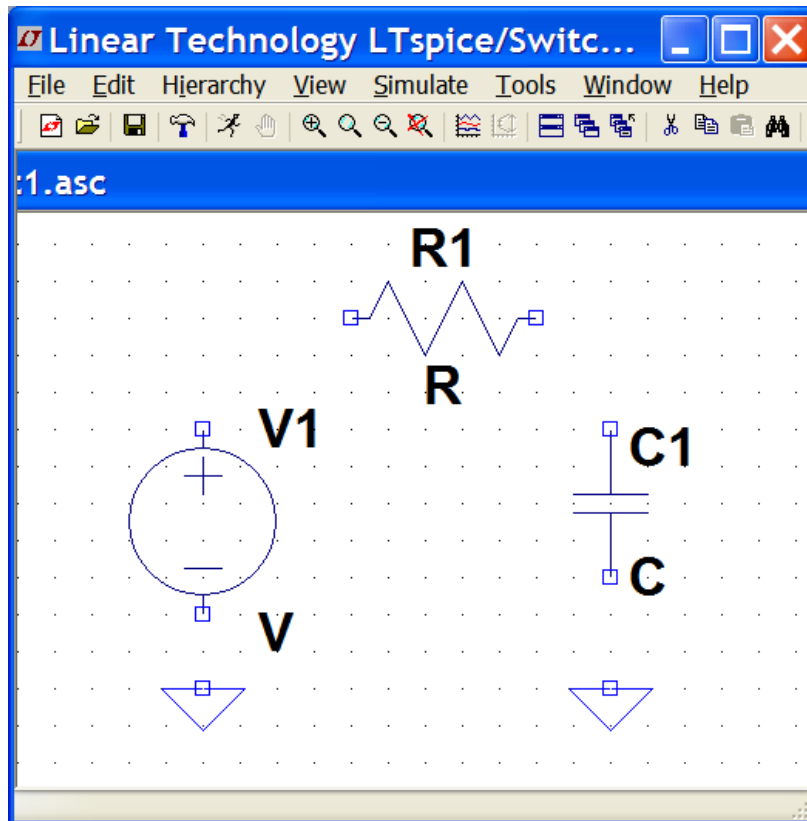
Hints

- ❖ Use **MEG (or meg)** to specify 10^6 , not *M*
- ❖ Enter **1** for 1 Farad, not *1F*

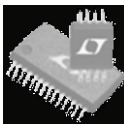
Wiring up a Simple RC Circuit



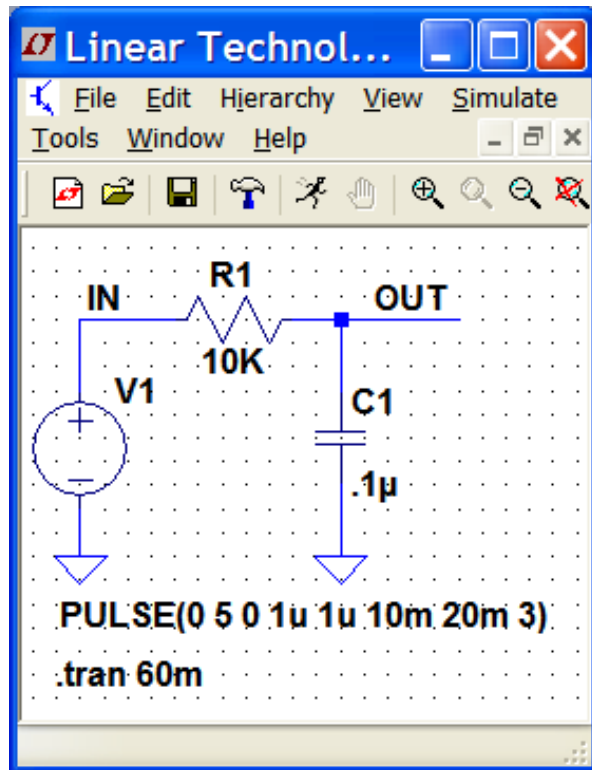
- ❖ Using the toolbar, select New Schematic
- ❖ Using the toolbar, select a Resistor, Capacitor and Ground. Place these on the schematic as shown below. Use **Ctrl R** to rotate before placement
- ❖ Using the toolbar, select Component. From the component window, type “voltage” in the dialog box, and click “OK” to place a voltage source



Wiring up a Simple RC Circuit



- ❖ Using the toolbar, select Wire. Wire up the RC circuit as shown below.
- ❖ Using the toolbar, select Label Net. Label the input/output nodes as shown below
- ❖ Right-Click on each component to change its value as shown below
- ❖ Right-Click on the voltage source and enter the parameters shown below under the “Advanced” tab.



Independent Voltage Source - V1

Functions

- ☐ (none)
- ☒ PULSE[V1 V2 Tdelay Trise Tfall Ton Period Ncycles]
- ☐ SINE[Voffset Vamp Freq Td Theta Phi Ncycles]
- ☐ EXP[V1 V2 Td1 Tau1 Td2 Tau2]
- ☐ SFFM[Voff Vamp Fcar MDI Fsig]
- ☐ PWL[t1 v1 t2 v2...]
- ☐ PWL FILE: Browse

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis[AC]

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

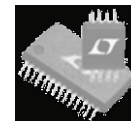
Make this information visible on schematic: ☒

Additional PwL Points

Make this information visible on schematic: ☒

Cancel OK

Independent Voltage Source - V1



Functions

- ☐ (none)
- ☒ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- ☐ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Voff Vamp Fcar MDI Fsig)
- ☐ PWL(t1 v1 t2 v2...)
- ☐ PWL FILE:

Vinitial[V]:

Von[V]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

Make this information visible on schematic: ☒

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis(.AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

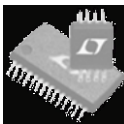
Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

Make this information visible on schematic: ☒

Editing Components



- ❖ Component attributes can be edited by pointing at the component with the mouse and Right-Clicking

Resistor - R6

Manufacturer:
Part Number:
Select Resistor
OK
Cancel

Resistor Properties

Resistance[Ω]: 10K
Tolerance[%]:
Power Rating[W]:

Inductor - L1

Manufacturer: Coilcraft
Part Number: D01608P-222
Select Inductor
OK
Cancel

Show Phase Dot ☐

Inductor Properties

Inductance[H]: 2.2u
Peak Current[A]: 2.3
Series Resistance[Ω]: 0.06
Parallel Resistance[Ω]: 55000
Parallel Capacitance[F]: 1.8p

Capacitor - Cp1

Manufacturer:
Part Number:
Type:
Select Capacitor
OK
Cancel

Capacitor Properties

Capacitance[F]: 22p
Voltage Rating[V]:
RMS Current Rating[A]:
Equiv. Series Resistance[Ω]:

- ❖ You can also edit the visible attribute and label by pointing at the text with the mouse and then right-clicking
 - ❖ Mouse cursor will turn into a text caret

Enter new Value for R6

Justification
Left
Vertical Text ☐
OK
Cancel

10K

Enter new Value for L1

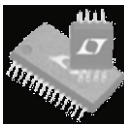
Justification
Top
Vertical Text ☐
OK
Cancel

2.2u

Enter new Value for Cp1

Justification
Left
Vertical Text ☐
OK
Cancel

22p



Component Database

- ❖ Components such as
 - ❖ Resistors, capacitors, inductors, diodes,
 - ❖ Bipolar transistors, MOSFET transistors, JFET transistors
 - ❖ Independent voltage and current sources
- ❖ You can access a database of known devices

Resistor - R6

Manufacturer: ----- OK

Part Number: ----- Cancel

Select Resistor

Resistor Properties

Select Standard Resistor

Quit and Edit Database OK

List All Resistors in Database Cancel

R[Ω]	Mfg.	Part No.	Power[W]	Tolerance[%]
10.00K			0.100	1.00
10.20K			0.100	1.00
9.76K			0.100	1.00
9.53K			0.100	1.00
10.50K			0.100	1.00
9.31K			0.100	1.00
10.70K			0.100	1.00
9.09K			0.100	1.00
11.00K			0.100	1.00
8.87K			0.100	1.00
11.30K			0.100	1.00
8.66K			0.100	1.00

Inductor - L1

Manufacturer: Coilcraft OK

Part Number: DD1608P-222 Cancel

Select Inductor

Show Phase Dot ☐

Inductor Properties

Select Stock Capacitor

Quit and Edit Database OK

List All Capacitors in Database Cancel

C[μF]	Mfg.	type	Part No.	Voltage[V]	Rser[Ω]
0.5	Nichicon	Al electrolytic	UPL1HR47MAH	50.0	3.900
0.5	Nichicon	Al electrolytic	UPR2AR47MAH	100.0	43.000
0.7	Nichicon	Al electrolytic	UPL1HR68MAH	50.0	3.700
1.0	TDK	X5R	C1608X5R1A10E	10.0	0.009
1.0	KEMET	X5R	C0603C105K8P	10.0	0.004
1.0	TDK	X7R	C3216X7R1C10E	16.0	0.007
1.0	AVX	Tantalum	TAJA105K016	16.0	11.000
1.0	KEMET	X7R	C0805C105K4R	16.0	0.031

Capacitor - Cp1

Manufacturer: ----- OK

Part Number: ----- Cancel

Type: -----

Select Capacitor

Capacitor Properties

Select Stock Capacitor

Quit and Edit Database OK

List All Capacitors in Database Cancel

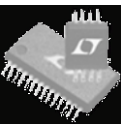
C[μF]	Mfg.	type	Part No.	Voltage[V]	Rser[Ω]
0.5	Nichicon	Al electrolytic	UPL1HR47MAH	50.0	3.900
0.5	Nichicon	Al electrolytic	UPR2AR47MAH	100.0	43.000
0.7	Nichicon	Al electrolytic	UPL1HR68MAH	50.0	3.700
1.0	TDK	X5R	C1608X5R1A10E	10.0	0.009
1.0	KEMET	X5R	C0603C105K8P	10.0	0.004
1.0	TDK	X7R	C3216X7R1C10E	16.0	0.007
1.0	AVX	Tantalum	TAJA105K016	16.0	11.000
1.0	KEMET	X7R	C0805C105K4R	16.0	0.031

Equiv. Parallel Capacitance[F]:

Mean Time Between Failures[hr]:

Parts Per Package:

How Do I Get Started using LTspice?



- ❖ Use one of the 100's demo circuit available on linear.com
 - ❖ Designed and Reviewed by Factory Apps Group
 - ❖ Go to <http://www.linear.com/software>
- ❖ Use a pre-drafted test fixture (JIG)
 - ❖ Provides a good starting point, but is not production-ready
 - ❖ Used to prove out part models, and are not complete designs.
 - ❖ Components are typically “ideal” components and will need to be modified based on your operating conditions
- ❖ Use the schematic editor to create your own design
 - ❖ LTspice contains models for most LTC power devices and many more
- ❖ Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
 - ❖ Also contains many very helpful discussion threads

LTspice Yahoo! User's Group Web Page

URL

LTspice : LTspice/SwitcherCAD III - Windows Internet Explorer

http://tech.groups.yahoo.com/group/LTspice/

File Edit View Favorites Tools Help

Y! Search Web Upgrade your Toolbar Now! Mail My Yahoo! Links

LTspice : LTspice/SwitcherCAD III

Yahoo! My Yahoo! Mail Search: Web Search

YAHOO! TECH Groups Sign In New User? Sign Up

Find your version of happiness on Yahoo! Personals. Search now.

I'M A W SEEKING A M AGE TO ZIP Search

LTspice · LTspice/SwitcherCAD III Search for other groups... Search

Home

Members Only Messages Post Files Photos Links Database Polls Members Calendar Promote

Info Settings

Group Information

Members: 11988 Category: Electrical Founded: Sep 27, 2002 Language: English

Already a member? Sign in to Yahoo!

Yahoo! Groups Tips

Did you know... Message search is now enhanced. find

Home

Activity within 7 days: 136 New Members - 87 New Messages - 14 New Files - New Questions

Description

Dedicated to the exchange of information about LTspice. LTspice/SwitcherCAD III (download now) is a complete and fully functional SPICE program that is available free of charge from Linear Technology.

Before using the software, please read the License Agreement found in the introductory LTspice Help pages.

Please report bugs directly to LTC. The program includes the correct address for that on its Help About dialog box (Menu command Help=> About LTspice/SwitcherCAD III). You will need to include any relevant files and a note on how to duplicate the problem.

For general questions, please read the program Help file, the group FAQ and try the group message Search before posting. If you post questions about a specific circuit, it is most helpful to also upload the schematic file (.asc) and any necessary symbol (.asy) and model files to the Files/Tmp folder. Never upload the big output file (.raw) of a simulation. Yahoo Groups allow you to easily correct or delete any upload mistakes.

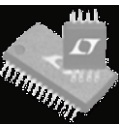
Hyperlinked lists of all uploaded files and links can be found in Files/ToC.

It is 'members only' to help avoid spammers. Please avoid personal attacks. The Group Moderators have no affiliation with Linear Technology.

Message History

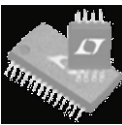
Join This Group!

Join the group here. As of Jan 2010, there are over 21,600 members!

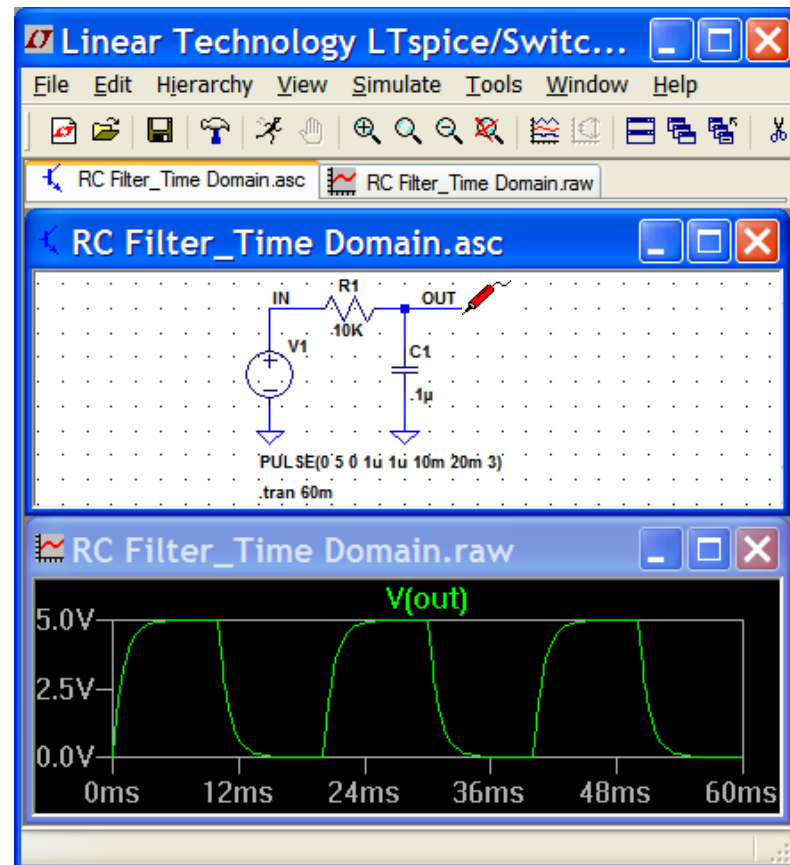
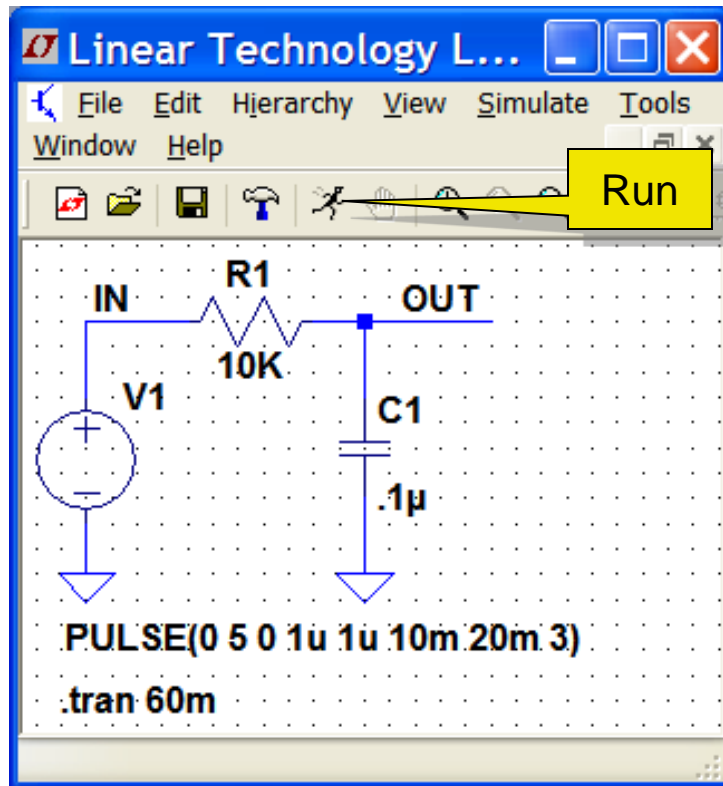


How Do You Run and Probe a Circuit in LTspice?

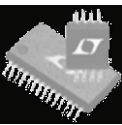
Running the RC Circuit Simulation



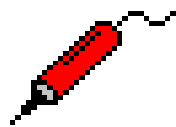
- ❖ With the RC circuit in the active window, click on the “Running Person” button on the tool bar
- ❖ The Edit Simulation Command window will appear. Set the Stop Time for 60msec, and click “OK”
- ❖ Using the mouse, click on the “OUT” node to display the output voltage waveform



Waveform Viewer



- ❖ LTspice has an integrated waveform viewer
- ❖ Plot the voltage on any wire by simply point and click

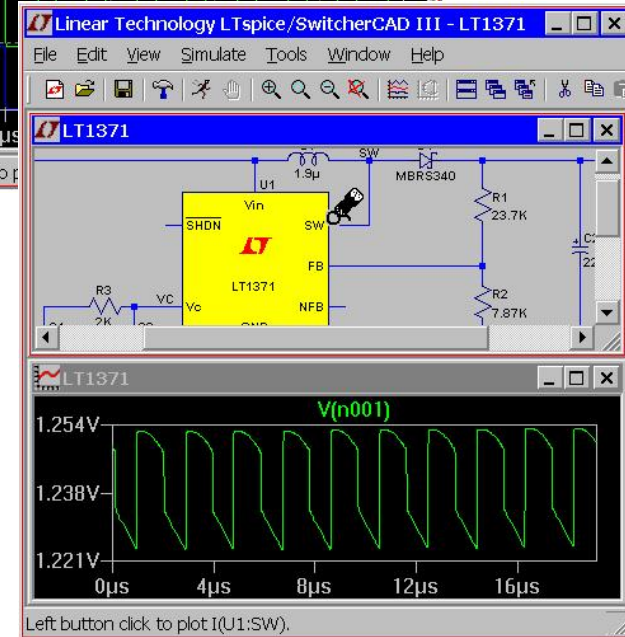
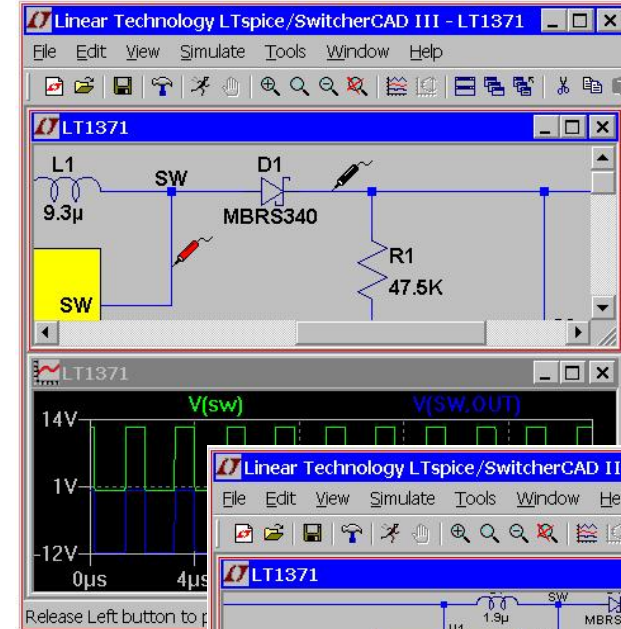


Voltage probe cursor

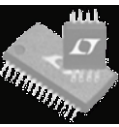
- ❖ Plot the current through any component with two connections by clicking on the body of the component






- ❖ R, C, L
- ❖ Convention of positive current is in the direction into the pin



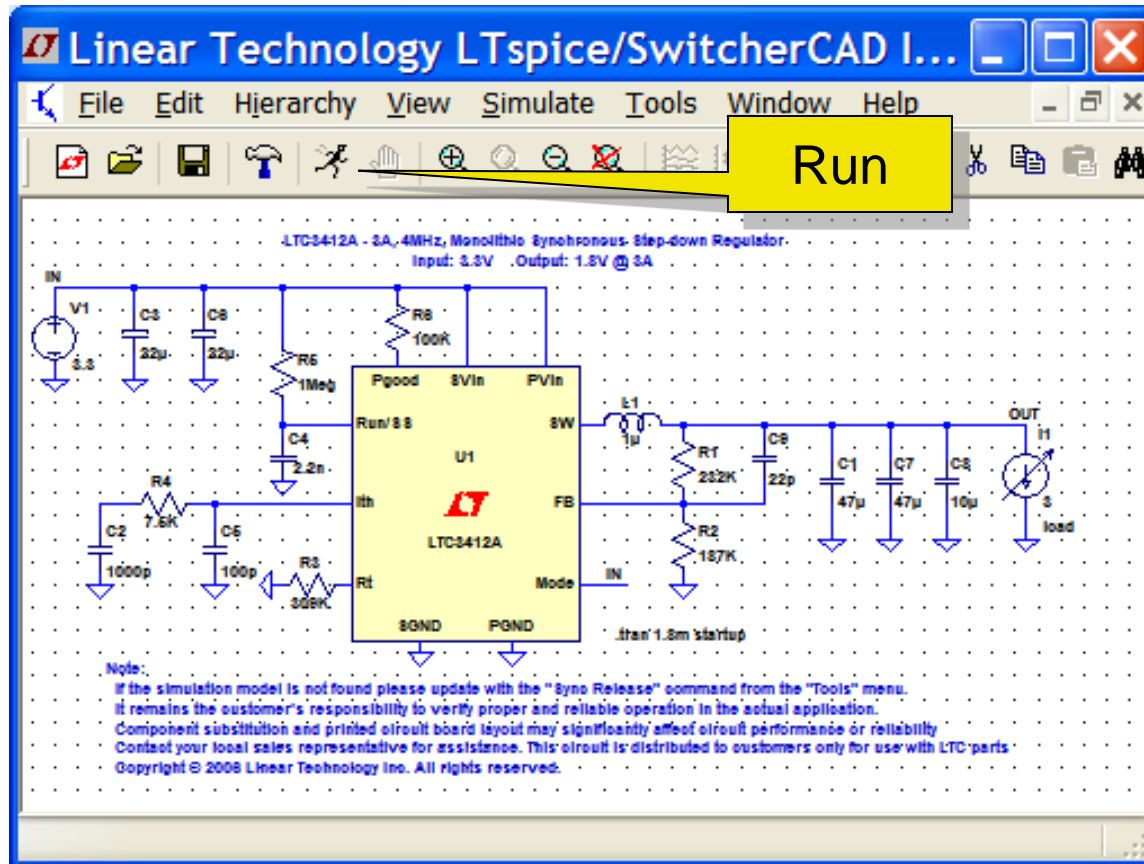
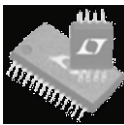
Running a Demo Circuit



- ❖ Access the LTC3412A demo circuit from the LTC website. It is located in the “LTspice Basic Lab Class” folder on your desktop
 - ❖ Click File ---> Open, and navigate to the LTspice Basic Lab Class folder on your desktop. Look for the file titled “LTC3412ADCLoad.asc”
- ❖ Hotlink Nomenclature:
 -  Class exercise
 -  Solution to exercise
 -  Circuits to explorer at your leisure

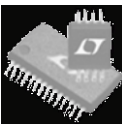


Running a Demo Circuit

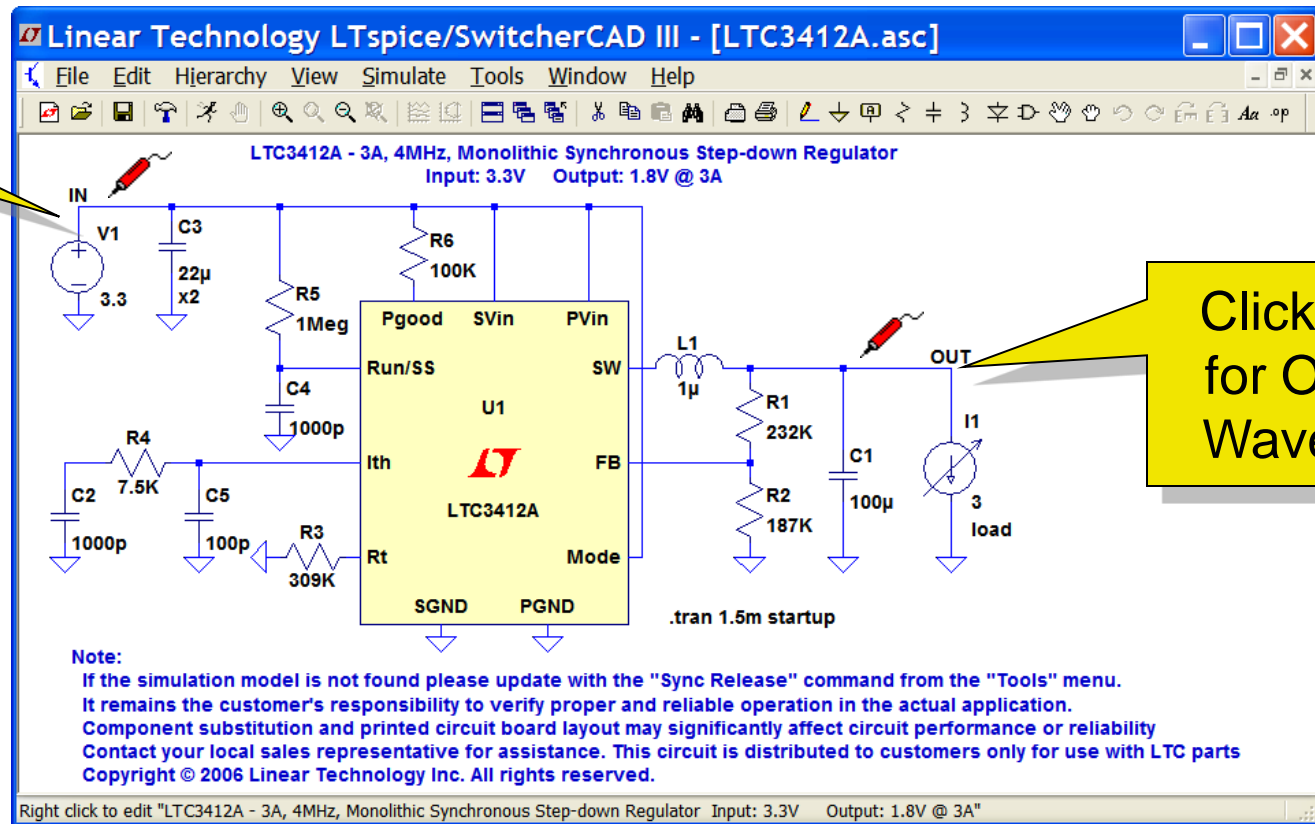


- ❖ Select the “Running Man” button on the toolbar
 - ❖ The Simulation will start and waveform window will open up
 - ❖ To view waveforms, please continue to the next page....

Probing a Demo Circuit



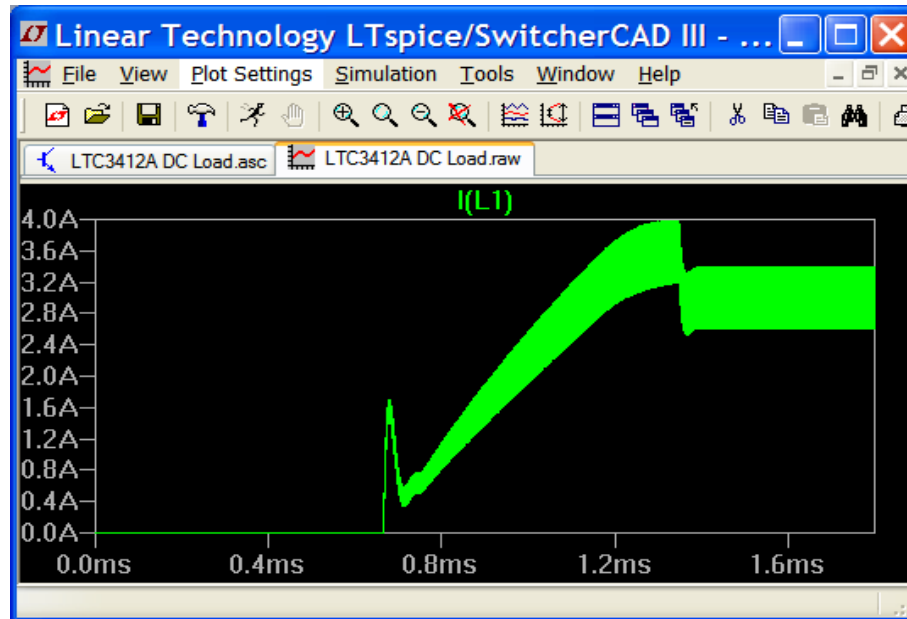
Click Here
for Input
Waveform



Click Here
for Output
Waveform

- ❖ All Demo Circuits have INs and OUTs clearly labeled to help you quickly select them
- ❖ Select the waveform of a node by clicking on IN and OUT

Zooming In and Out on a Waveform

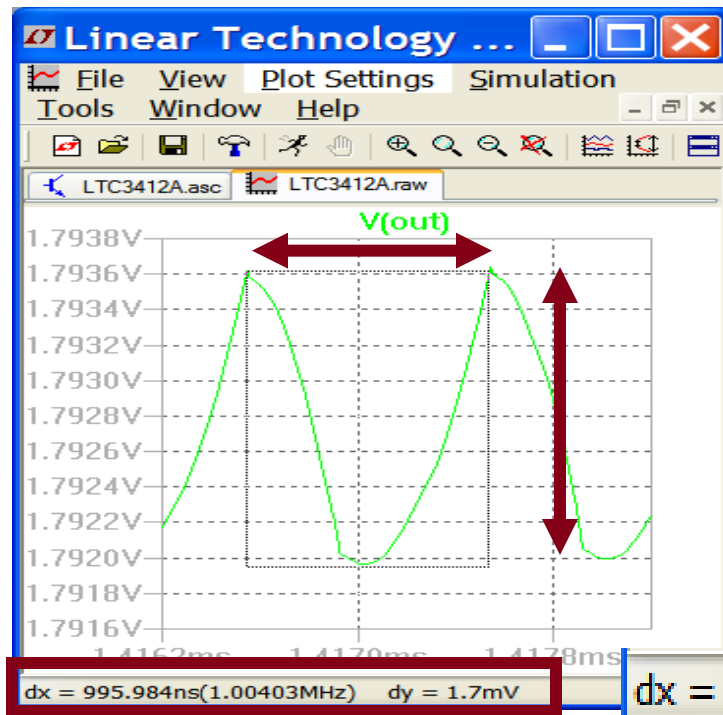


- ❖ Using the mouse, click on inductor L1 to display the inductor current waveform
- ❖ In the waveform window, use the mouse to zoom in and out
 - ❖ Click and drag a box about the region you wish to see drawn larger
- ❖ Using the toolbar, click on “Zoom full extents”, to zoom back out

Measuring V, I and Time in the Waveform (Measurement Using Zoom)

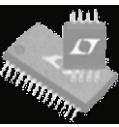


1. Drag a box about the region you wish to measure
 - ❖ Left-Click, drag, and hold
2. View the lower left corner of the window for the status bar. The dx and dy measurement data is displayed here.
3. Use Undo from the File menu or press “F9”



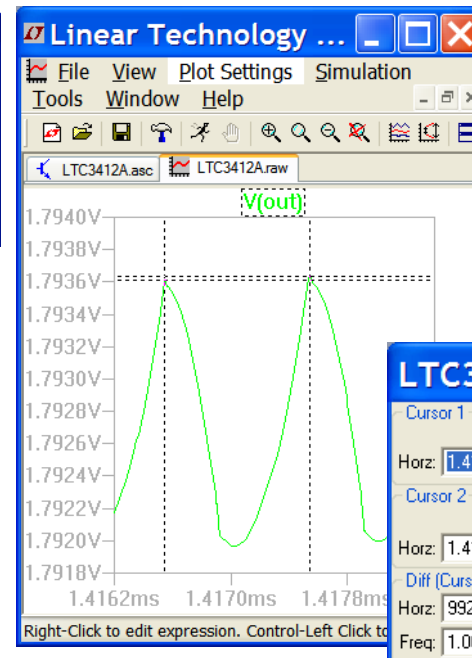
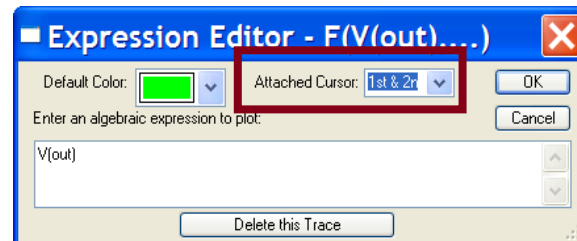
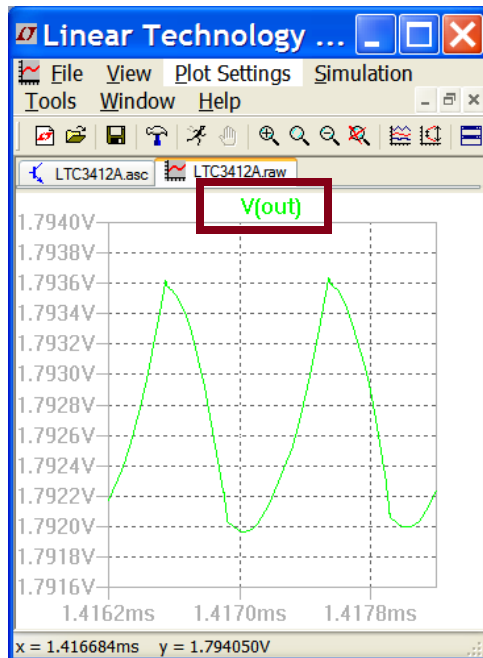
$dx = 995.984ns(1.00403MHz)$ $dy = 1.7mV$

Measuring V, I and Time in the Waveform (Measurement Using Cursors)

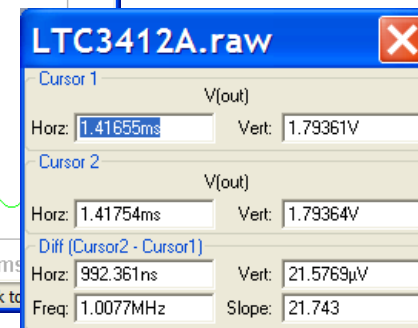


1. Right-Click on the waveform name in the waveform window
2. For “Attached Cursor”, select “1st & 2nd”
3. Position cursors to make desired measurements.

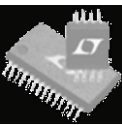
1. → 2. → 3.



Result



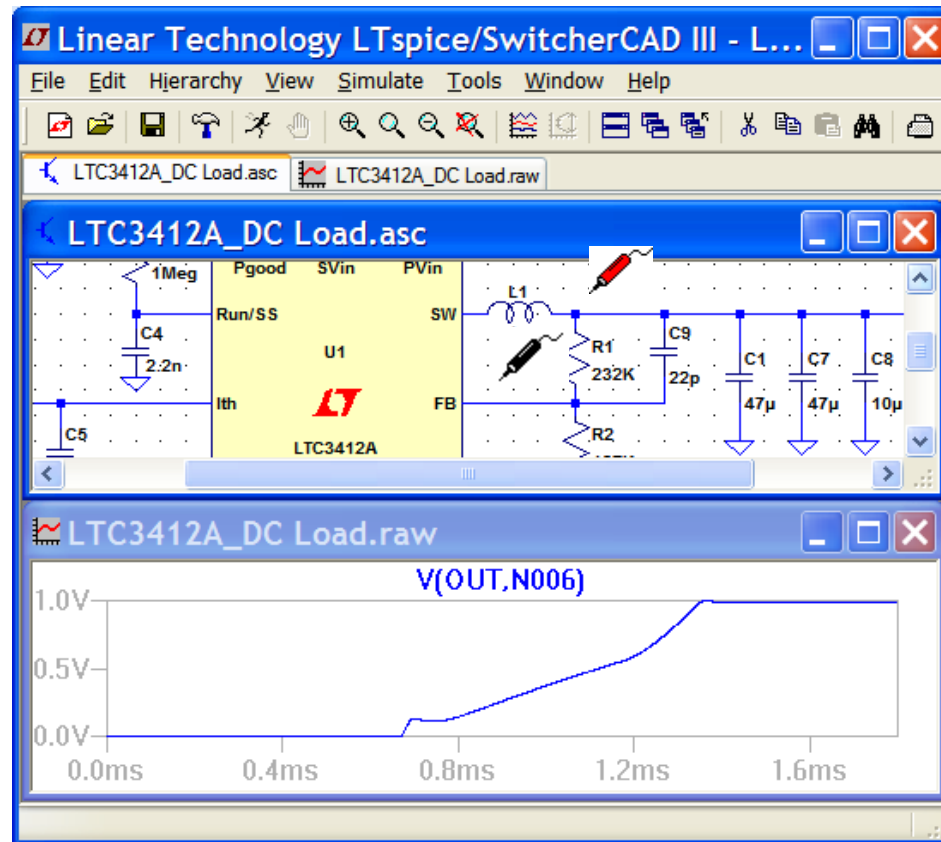
Differential Voltage Measurement



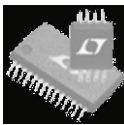
- ❖ Click on one node and drag the mouse to another node
 - ❖ Red voltage probe at the first node
 - ❖ Black probe on the second
- ❖ Will produce a differential voltage measurement

Example:

**Measure across
LTC3412A top
resistor in
feedback divider**



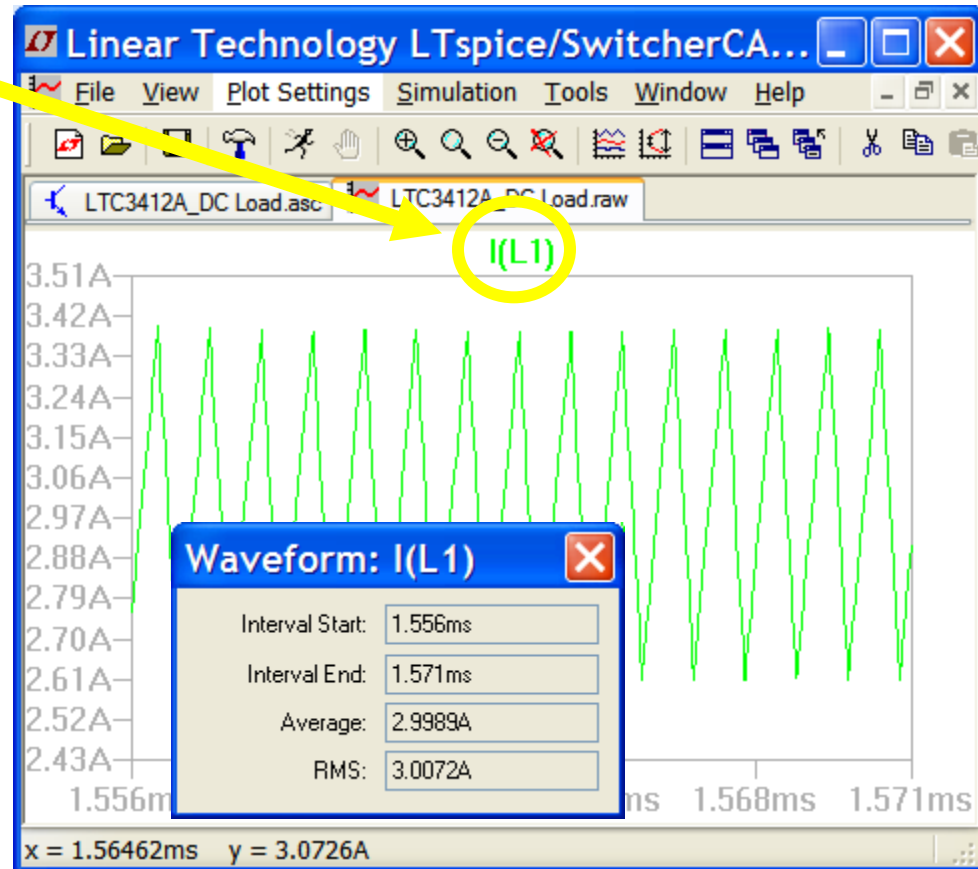
Average & RMS Calculations



- ❖ Average & RMS Current, Voltage, or Power Dissipation
- ❖ Click on inductor L1 to display the inductor current waveform
 - ❖ **Ctrl-Left-Click** the I(L1) trace label in the waveform view

Example:

Measure average and RMS current for inductor in LTC3412A circuit. Zoom in as shown for this waveform.



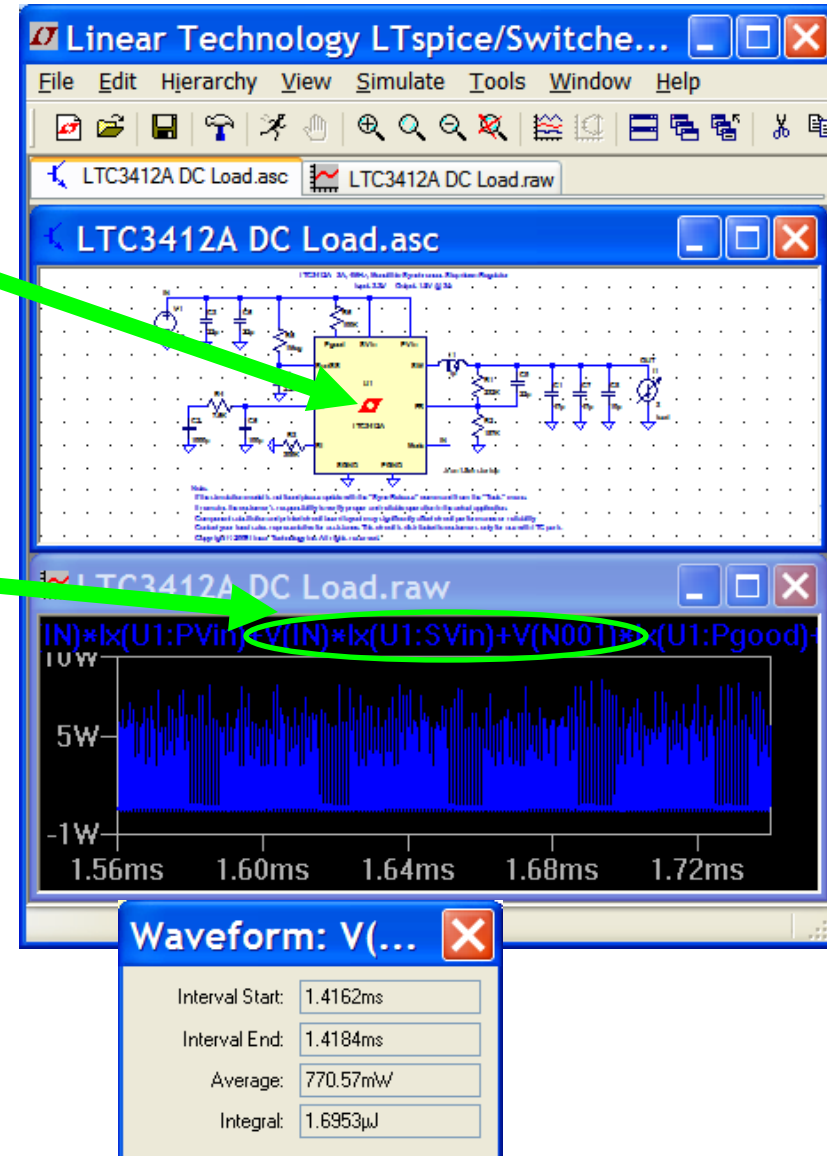
Instantaneous & Average Power Dissipation



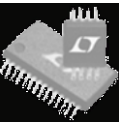
- ❖ Instantaneous Power Dissipation
 - ❖ **Hold down the Alt key and Left-Click** on the symbol of the LTC3412A
 - ❖ Waveform is displayed in units of Watts
- ❖ Average Power Dissipation
 - ❖ **Click, hold, and drag** in the waveform window to display waveform at steady state
 - ❖ **Ctrl-Left-Click** on the Power Dissipation Trace Label in the waveform view
 - ❖ Waveform summary window will appear which shows power dissipation in the IC

Example:

Measure the power dissipation in the LTC3412A IC

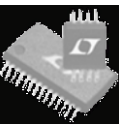


Advantages of Labeling

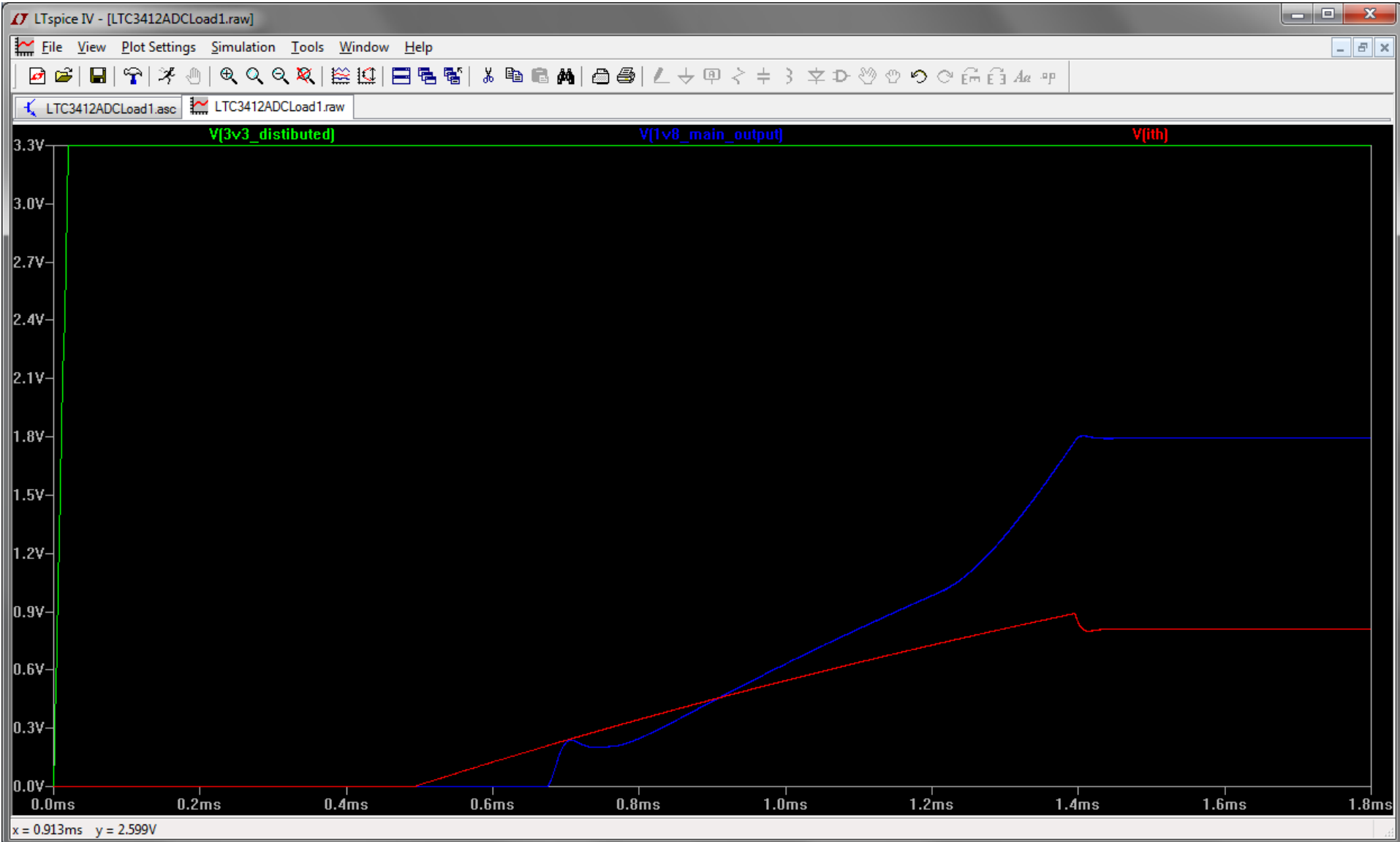


- ❖ Replaces arcane SPICE machine node names with easy to understand and remember human names
- ❖ Allows LTspice circuit nodes to match those on your production schematic, i.e. “TP15”

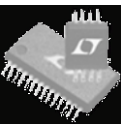
Advantages of Labeling



❖ Compare this....

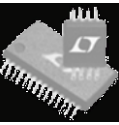


Advantages of Labeling



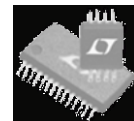
❖ To this....



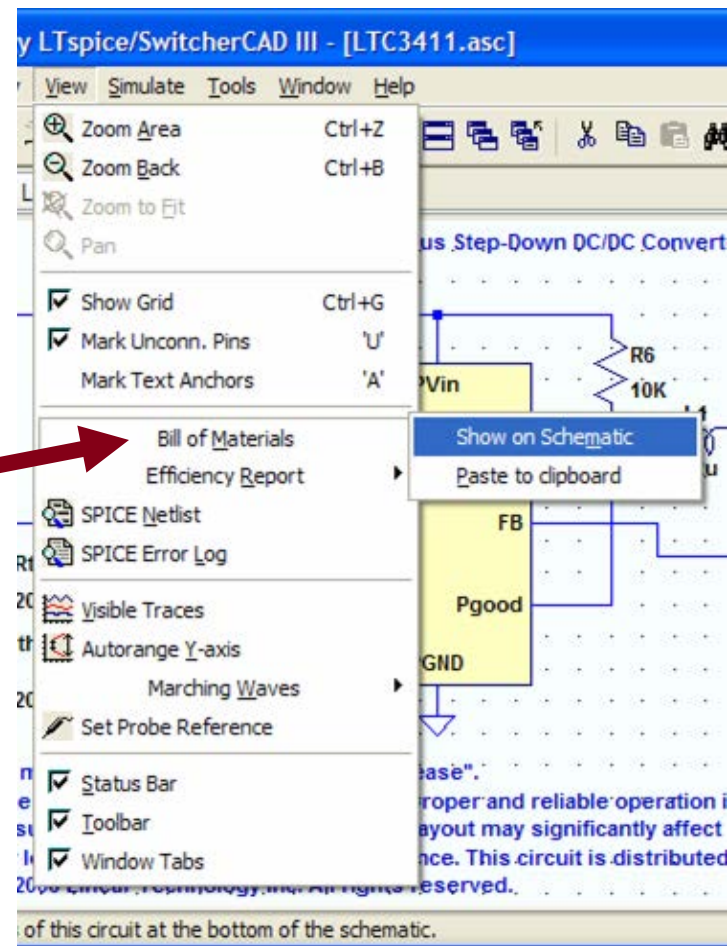
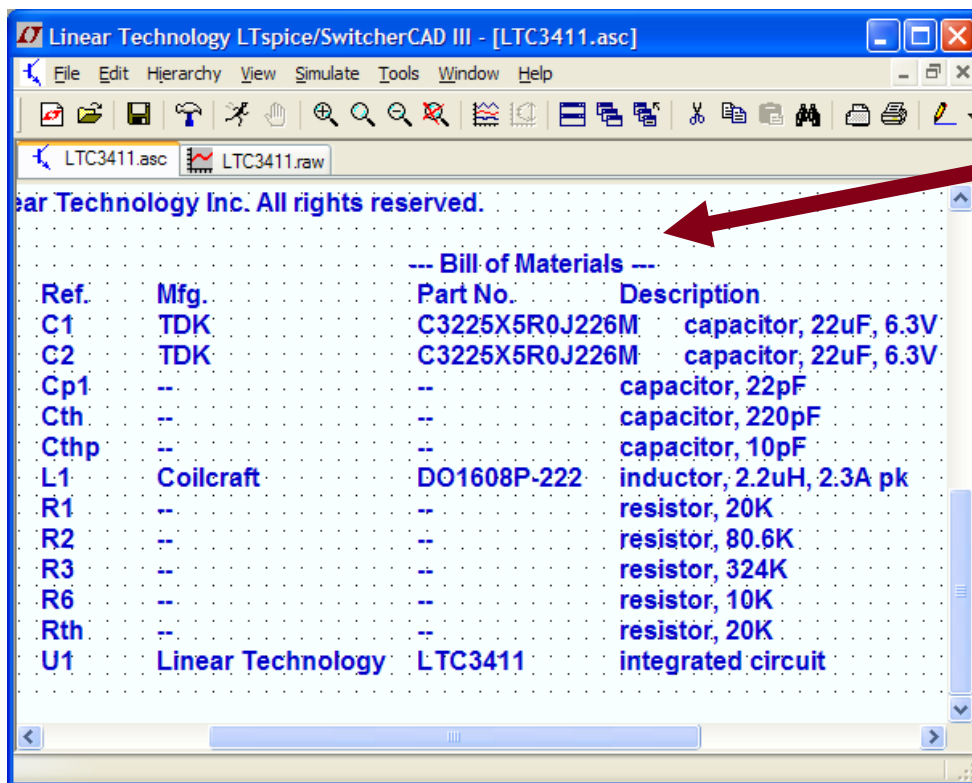


Generating a BOM and Efficiency Report

BOM



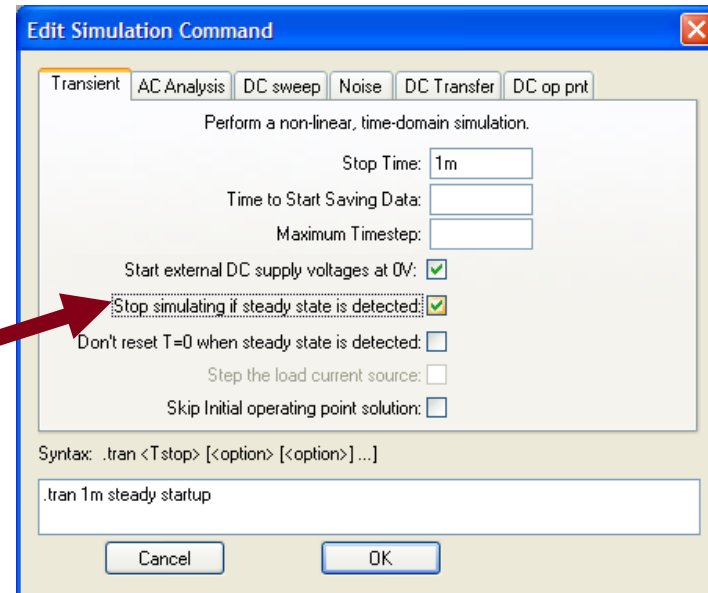
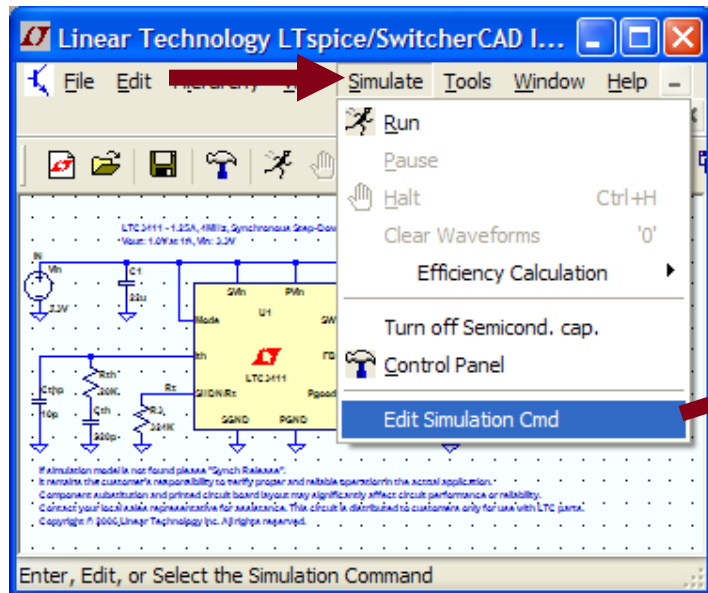
- ❖ Under View select Bill of Material
 - ❖ Displayed on Diagram
 - ❖ Paste to Clipboard



Computing Efficiency & Dissipation

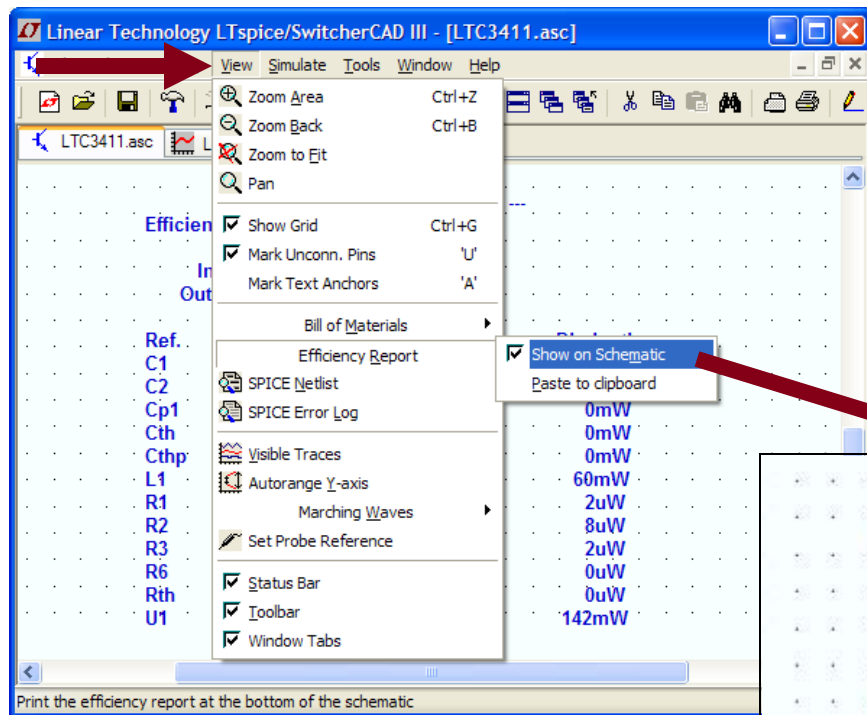
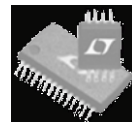


- ❖ To compute efficiency of SMPS circuits:
 - ❖ Check the "Stop simulating if steady state is detected" on the Edit Simulation Command editor
 - ❖ Rerun simulation
 - ❖ Use the menu command View=>Efficiency Report



Automatic detection of steady state may not always work – criteria for steady state detection may be too strict or too lenient

Viewing Efficiency Report

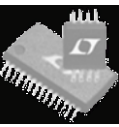


--- Efficiency Report ---

Efficiency: 83.1%

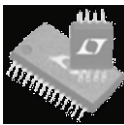
Input: 1.2W @ 3.3V
Output: 997mW @ 999mV

Ref.	I _{rms}	I _{peak}	Dissipation
C1	0mA	0mA	0mW
C2	99mA	177mA	0mW
Cp1	0mA	0mA	0mW
Cth	0mA	0mA	0mW
Cthp	0mA	0mA	0mW
L1	1003mA	1176mA	60mW
R1	0mA	0mA	2uW
R2	0mA	0mA	8uW
R3	0mA	0mA	2uW
R6	0mA	0mA	0uW
Rth	0mA	0mA	0uW
U1	1003mA	1176mA	142mW

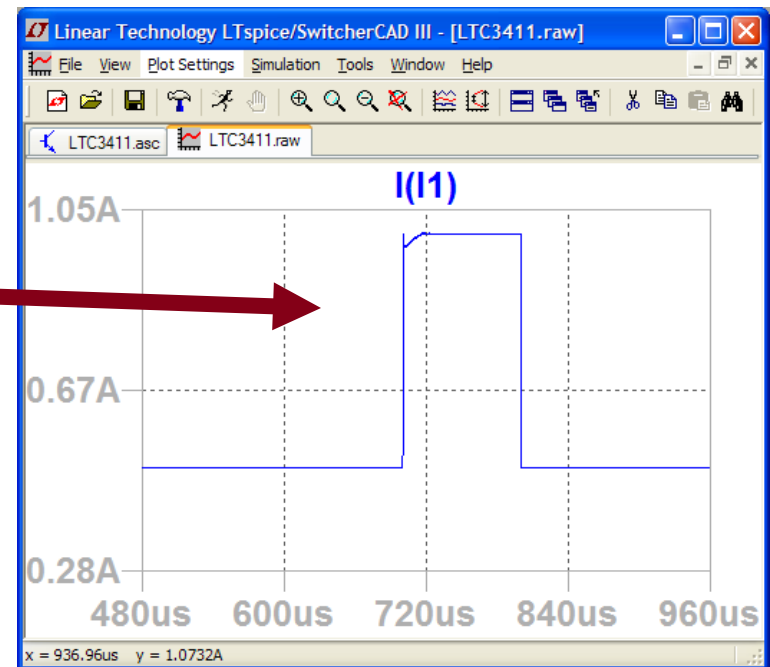
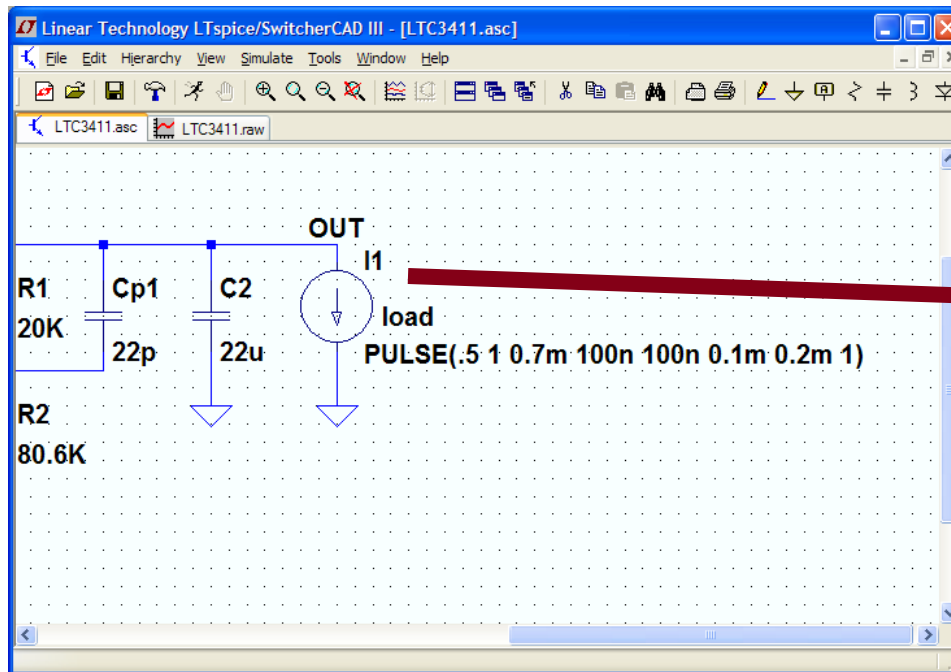


Simulating a Transient Response

Current Load and Pulse Function



- ❖ You can simulate a load with a Resistor or Current load
- ❖ In particular the Pulse function in a current load is helpful in transient response analysis
 - ❖ Steps a current load from one value to another value



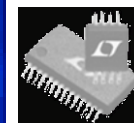
Edit the Current Load to a Pulse Function



- ❖ In the LTC3412A simulation, Right-Click on the current load
- ❖ Select “Pulse”
- ❖ Modify the Attributes (see next slide). Click “OK”.



Independent Current Source - I1



Functions

- ☐ (none)
- ☒ PULSE(I1 I2 Tdelay Trise Tfall Ton Period Ncycles)
- ☐ SINE(Ioffset Iamp Freq Td Theta Phi Ncycles)
- ☐ EXP(I1 I2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Ioff Iamp Fcar MDI Fsig)
- ☐ PWL(t1 i1 t2 i2...)
- ☐ TABLE(v1 i1 v2 i2...)

I1[A]:

1

I2[A]:

3

Tdelay[s]:

1.4m

Trise[s]:

1u

Tfall[s]:

1u

Ton[s]:

100u

Tperiod[s]:

200u

Ncycles:

2

Additional PwL Points

Make this information visible on schematic: ☒

DC Value

DC value: 3

Make this information visible on schematic: ☒

Small signal AC analysis(AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

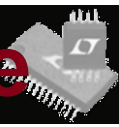
This is an active load: ☒

Make this information visible on schematic: ☒

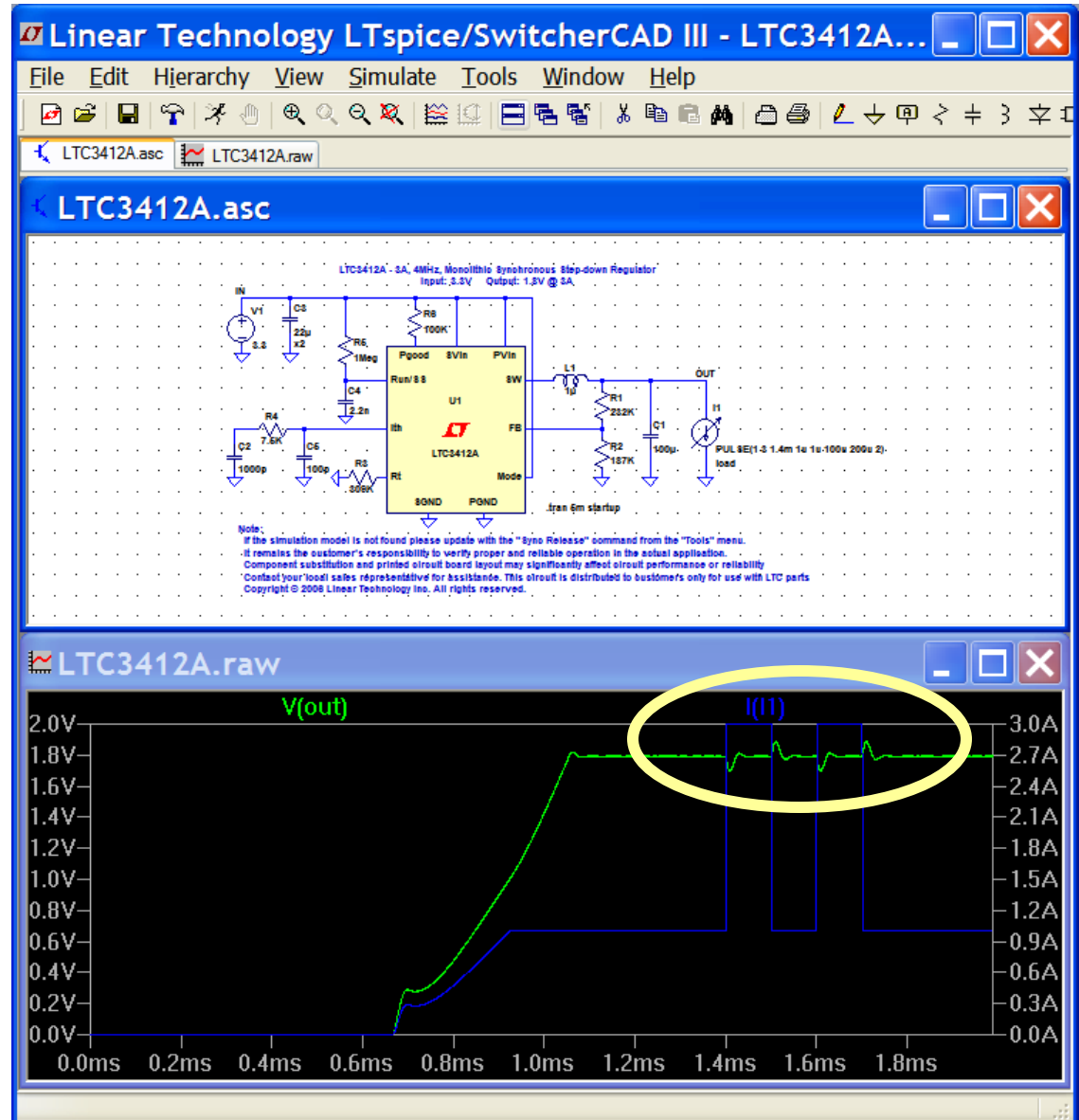
OK

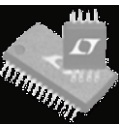
Cancel

Run the Simulation for Transient Response

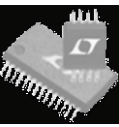


- ❖ Run the simulation
- ❖ Click on the OUT node to display V_{out}
- ❖ Click on the output current load to display I_{out}
- ❖ Notice the presence of the pulse load



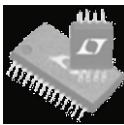


AC Analysis



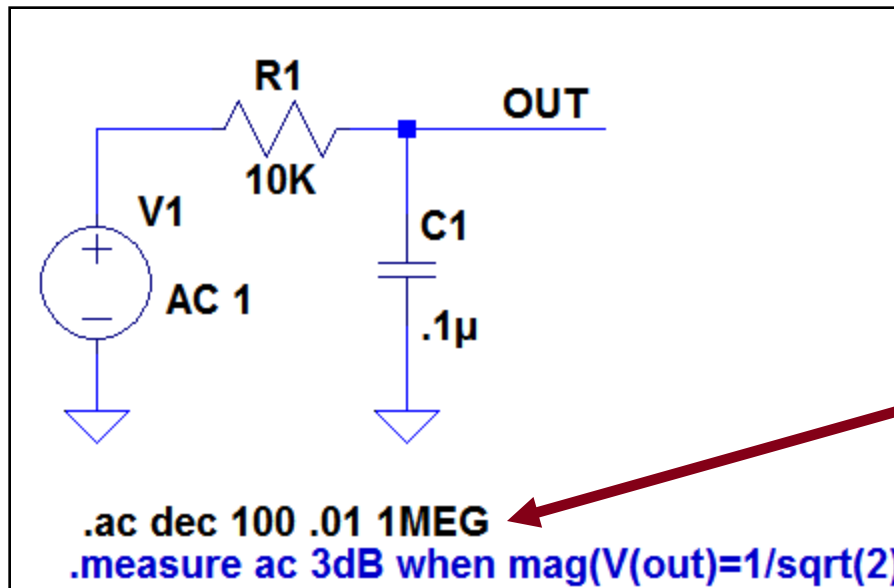
AC Analysis Overview

- ❖ Performs small signal AC analysis linearized about the DC operating point
- ❖ Useful for analysis of filters, networks, stability analysis, and noise considerations



Simulating AC Analysis – RC Filter

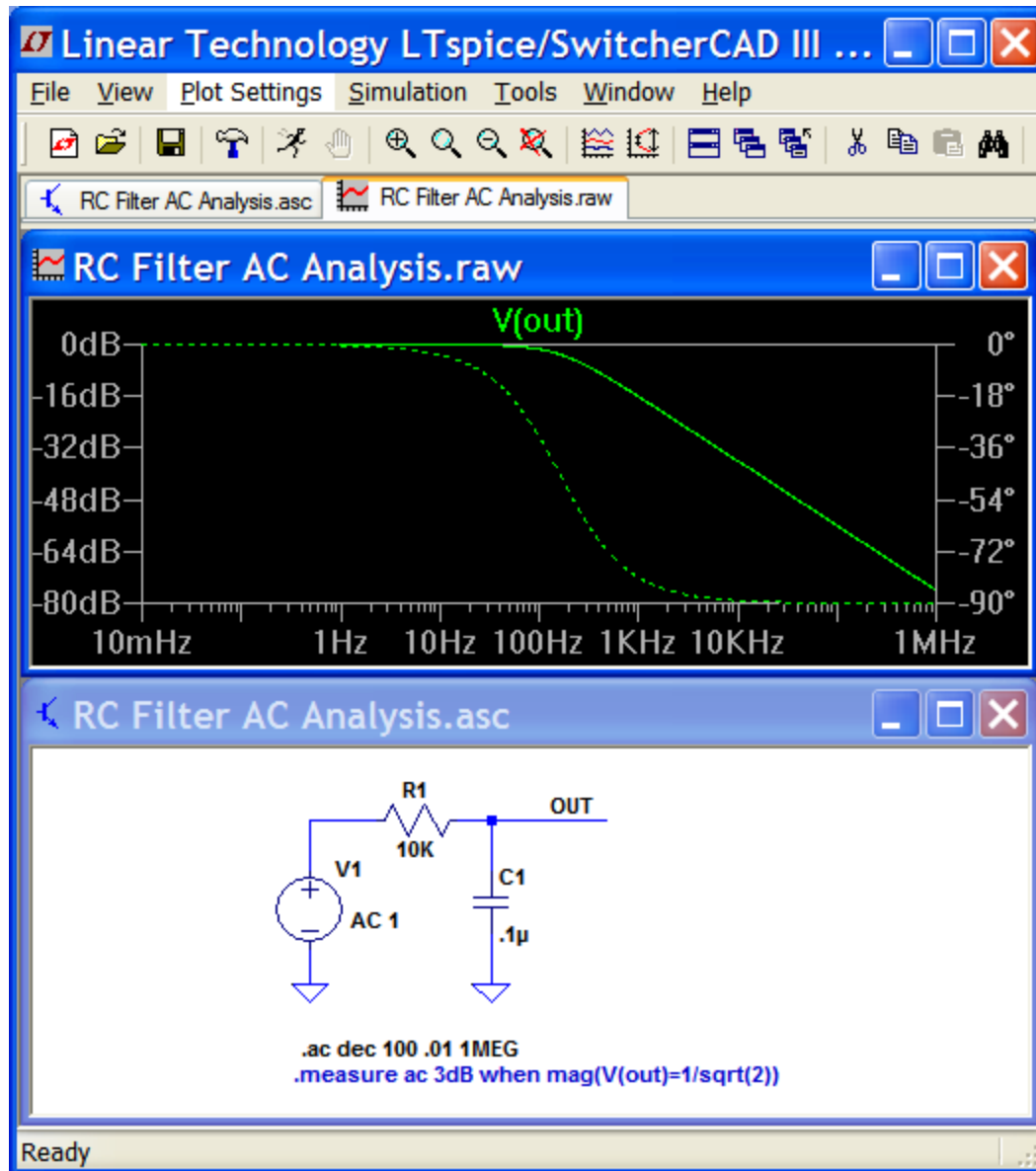
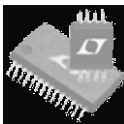
- ❖ Single pole filter using RC network
- ❖ Syntax: `.ac <oct, dec, lin> <Nsteps> <StartFreq> <EndFreq>`
- ❖ Example: RC network and `.ac dec 100 .01 1MEG`



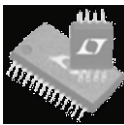
-3dB point:
 $1/(2 \cdot \pi \cdot R \cdot C) = 159\text{Hz}$

**Right-click on .tran
command and select
“AC Analysis”**

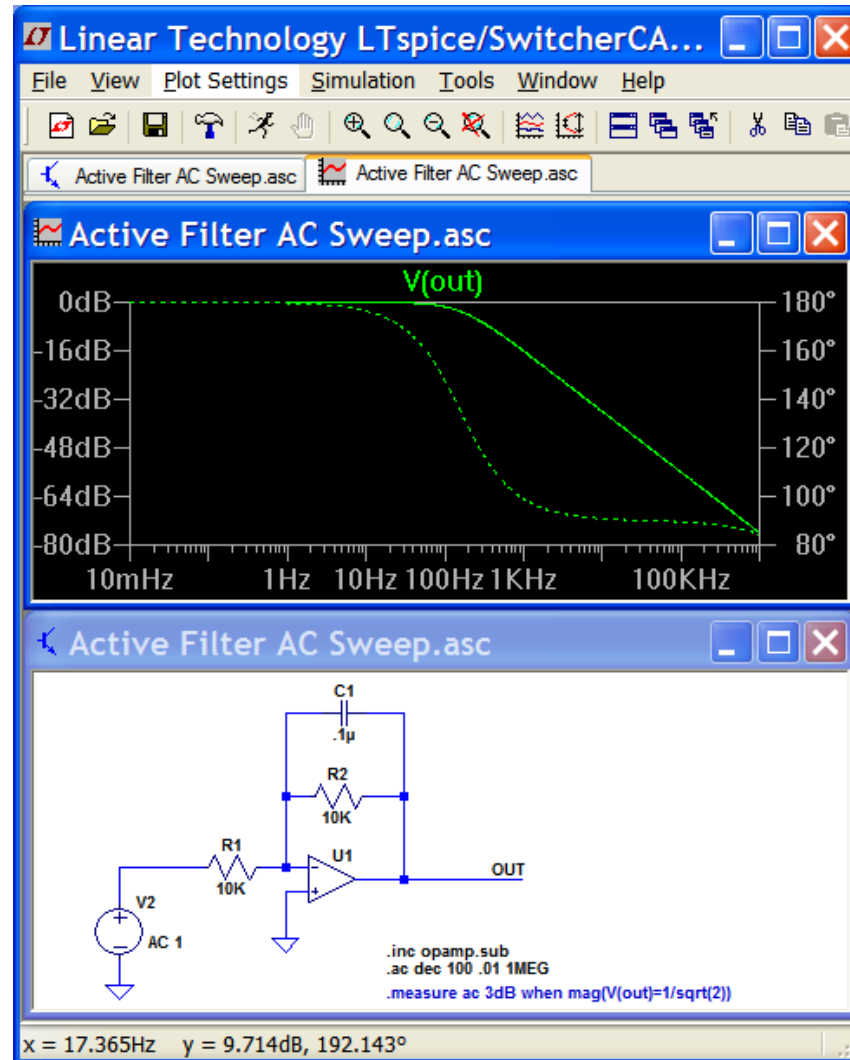
Simulating AC Analysis – RC Result

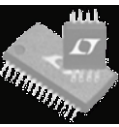


Simulating AC Analysis – Active Filter



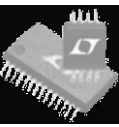
- ❖ Single pole active filter using an opamp





Importing Third-Party SPICE Models

Importing Third-Party Spice Models

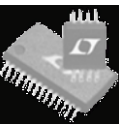


To import a third party spice model:

- 1.) Download the spice model file from the manufacturer's website
- 2.) Make sure that the spice model file is located in the same directory as the LTspice simulation file
- 3.) Add the following spice directive to the LTspice simulation file (Edit ---> SPICE Directive):

```
.include spice_model_file_name.abc
```
- 4.) Modify the component name in the LTspice schematic to match the component name contained in the spice model file (Right-Click on the device name, and modify accordingly)

Importing Third-Party Spice Models



The following items are CRITICAL!

- 1.) The file name in the .include statement must match the spice model file name identically (except for case)! The file name syntax can be anything, just make sure that all of the characters match.
- 2.) The model name in the spice model file must match the device name in the LTspice schematic identically! The model name syntax can be anything, just make sure that all of the characters match.

Importing Third-Party Spice Models



Spice Model Example #1:

1N5244B.mod - Notepad

```
File Edit Format View Help
* 1N5244B Zener Diode
*
.MODEL 1N5244B1 D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

File name = 1N5244B.mod

Model name = 1N5244B1

Summary: The file and model names are irrelevant. Just make sure that the LTspice simulation device name and .include file name match those of the spice model file.

Spice Model Example #2:

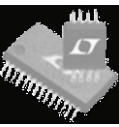
Joe.txt - Notepad

```
File Edit Format View Help
* 1N5244B Zener Diode
*
.MODEL Everest D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

File name = Joe.txt

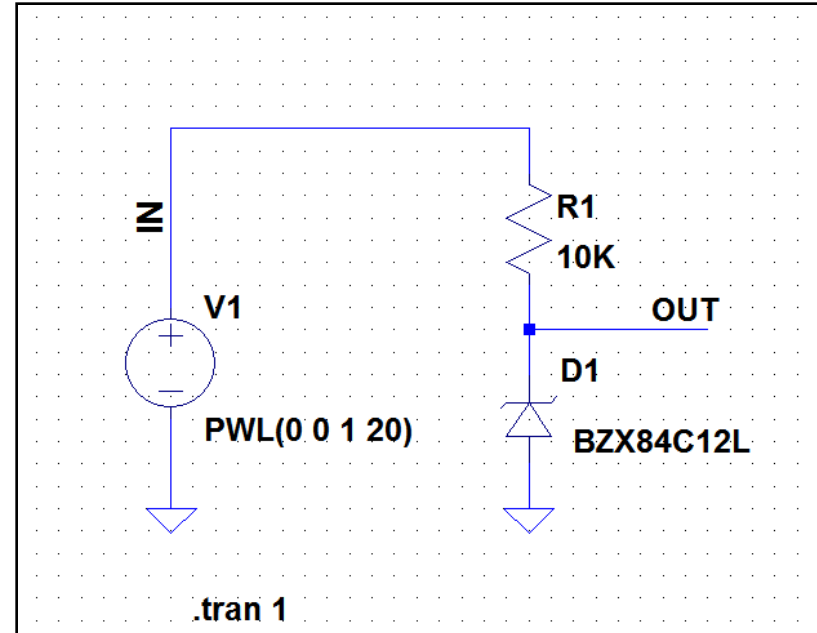
Model name = Everest

Importing Third-Party Spice Models



Hands-on Exercise:

- 1.) Navigate to the LTspice Basic Lab Class folder
- 2.) Open up the simulation file titled "ZenerImportExample.asc"
- 3.) Open up the SPICE model file titled "1N5244B.txt" and note the device model name.
- 4.) Modify the simulation file so that it uses the 1N5244B third-party SPICE model based on the instructions provide on the previous slides
- 5.) Run the simulation and probe the IN and OUT nodes

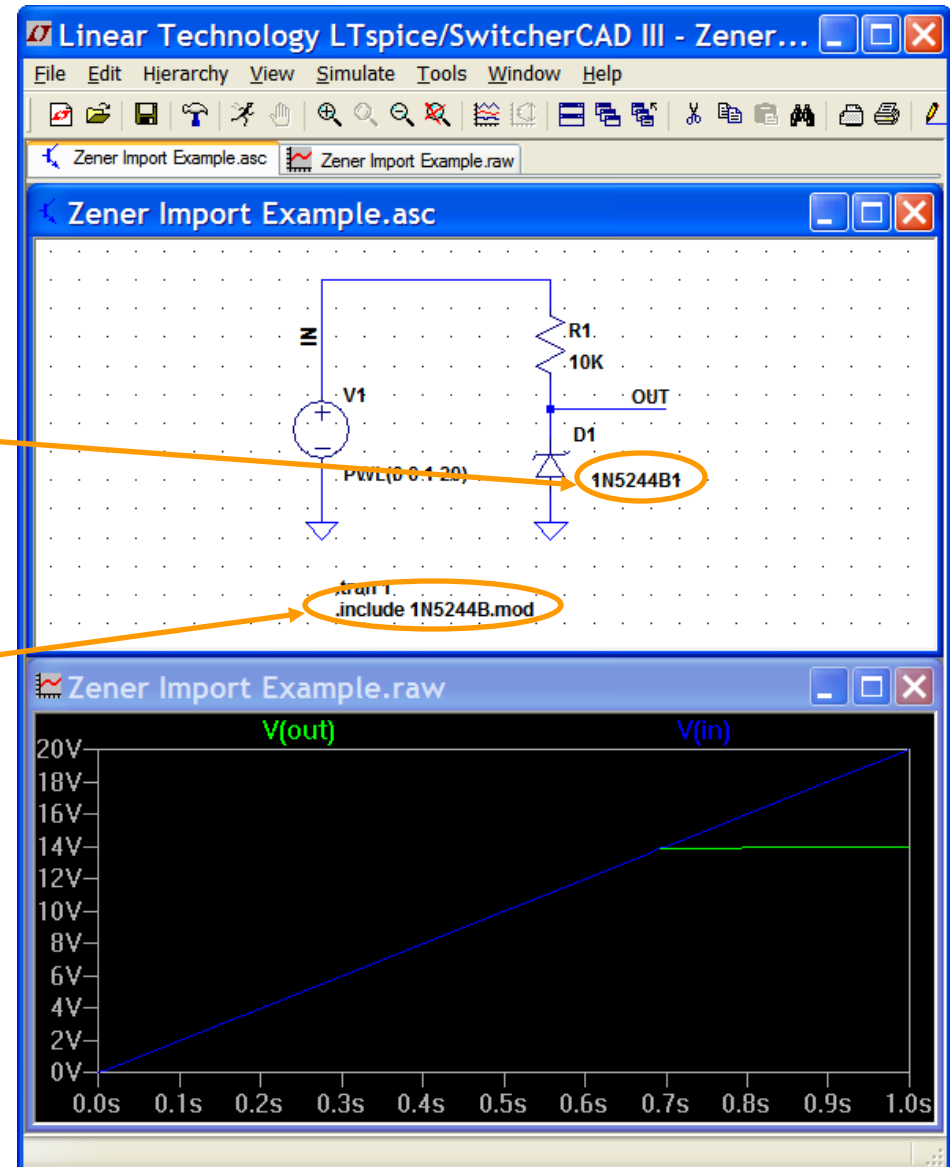


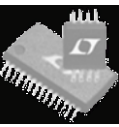
Importing Third-Party Spice Models



Solution:

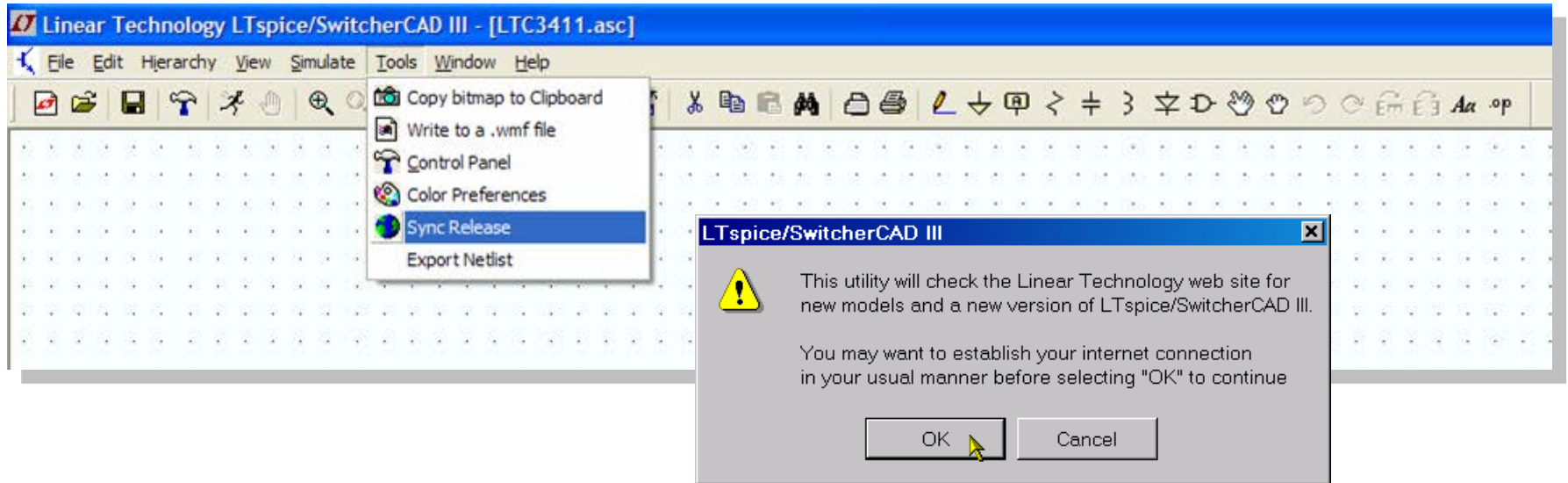
- 1.) Zener name changed to 1N5244B1 to match model name in the SPICE model file. Right-Click on the diode name text to change.
- 2.) .include SPICE directive added to link to the SPICE model file. Use the Edit pulldown menu ---> Spice Directive to add this SPICE directive to your simulation.
- 3.) Result after clicking on the Running Person symbol on the toolbar and probing the IN and OUT nodes.





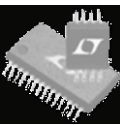
More Information and Support

Reminder to Periodically Synch Release



- ❖ It is important to sync your release of LTspice once a month to get the latest updates
 - ❖ Software update and bug fix
 - ❖ Models
 - ❖ Sample circuits and examples
- ❖ Vista users
 - ❖ You must “Run as administrator” scad.exe or its shortcut even if you are logged in as an administrator

Built-in Help System



The screenshot shows the **scad3** software interface. The **Help Topics** window is open, displaying a tree view on the left with categories like **Introduction**, **Modes of Operation**, **Schematic Capture**, **Editing Components**, **Waveform Viewer**, **LTspice**, and **Transient Analysis Options**. The **Schematic Colors** topic is selected. The main text area explains that the **Tools=>Color Preferences** command allows users to set colors for schematics using red, green, and blue sliders. A **Color Palette Editor** dialog is overlaid on the text, showing a sample schematic with components like **C1** (100p), **C2** (100p), **R1** (5K), **LBO**, **Vc**, and **LBI**. The dialog includes a **Selected Item** dropdown set to **Wires** and a **Selected Item Color Mix** section with sliders for Red (0), Green (0), and Blue (255). Buttons for **OK**, **Cancel**, **Apply**, and **Defaults** are at the bottom.

Schematic Colors

The menu command **Tools=>Color Preferences** colors allows you to set the colors used in displaying the schematics. You click on an object in the sample schematic and use the red, green and blue sliders to adjust the colors to your preferences.

Color Palette Editor

Click on an item above to change its color.

Selected Item: Wires

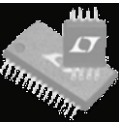
Selected Item Color Mix:

Color	Value
Red	0
Green	0
Blue	255

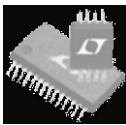
Buttons: OK, Cancel, Apply, Defaults

Note: Non-electrical graphical annotations made to schematics such as lines and circles will be draw in the same color as a component body.

PDF User's Guide



- ❖ Download the PDF User's Guide Manual:
 - ❖ <http://LTspice.linear.com/software/scad3.pdf>



Appendix A – Summary of Special Mouse and Keyboard Commands

Schematic-Based Special Commands:

1. Alt-Left-Click on a wire
 - ❖ This will display the waveform for the current flowing in the wire
2. Alt-Left-Click on a component
 - ❖ This will display the instantaneous power dissipation in the component
3. Ctrl-Right-Click on a component
 - ❖ Allows you to edit embedded component attributes

Waveform-Based Special Commands:

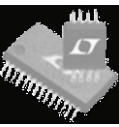
1. Ctrl-Left-Click on a waveform title
 - ❖ Displays the average and RMS values for the waveform
2. Left-Click on node and drag to another node
 - ❖ Displays differential voltage

Appendix B – Summary of Additional Features



1. To pause a simulation:
 - ❖ “Simulate” pull down menu ---> Pause
 - ❖ There is no toolbar button for this function
2. To zoom in/out using the schematic editor:
 - ❖ Just use the wheel on your mouse
3. To pan around a schematic
 - ❖ Left-Click the mouse and hold, then drag
 - ❖ Tilt wheel to move right and left

Thank you for attending, and happy simulating!



Homework: Once you return to the office, go back over the training materials within a week!

For More Information:

Existing Arrow Customers: 800 777 2776

New Customers: 800 833 3557

www.arrowna.com/powermanagement